COMPUTER SIMULATION FOR FINDING OPTIMUM GATELOCATION IN PLASTIC INJECTION MOULDING PROCESS

B. KISHORE KUMAR[1], K MADHU SREE[2], K.TRINATH KUMAR[3]
1PG Research scholor, 2Asso.Professor, 3Individual researcher
[1]Mechanical Engineering Department
[1]INTEL Engineering College, Anantapur, AP ,*INDIA*.

Abstract : Due to significant demand in plastic product, plastic industries area unit growing during a quickest rate. Plastic injection moulding begins with mould creating in producing of advanced shapes. The optimum gate location is one in every of the foremost vital criterions in mould style. Mould Flow analysis may be a powerful simulation tool to optimize the gate location and to predict the assembly time needed at rock bottom potential price. Verification mistreatment simulation needs a lot of less time to attain a top quality result, and with no material prices, as compared with the standard trial-and-error ways on the assembly floor.
In the gift work, Injection mould method is taken as a main concern and head lightweight cowl of a Alto automobile is factory-made by variable range of gate locations. lightweight ought to have the duvet intermittently, so, there mustn't be any flaws like blow holes, sink marks or weld lines, throughout material filling within the mould. As a locality of the work Injection Mould input pressure is varied to attenuate the mould flow defects and to see the optimum gate location. Mould Flow Plastic authority simulation tool in Pro/Engineer is employed for the analysis and also the optimum gate location is found with least defects.

Keywords: Injection moulding; Mould Flow Plastic authority (MPA); Gate location; moulding defects

I. INTRODUCTION

Injection molding is that the most typically used producing method for the fabrication of plastic elements. a large type of product area unit factory-made mistreatment injection molding, that vary greatly in their size, complexity, and application. The injection molding method needs the utilization of associate injection molding machine, raw plastic material, and a mold. The plastic is melted within the injection mildewing machine and so injected into the mold, wherever it cools and solidifies into the ultimate half.

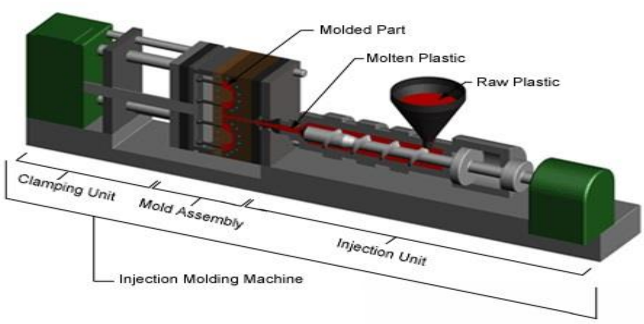

Fig. 1.1 Injection Moulding Machine Model

	Babyplast	Powerline	Maxima
Clamp force (ton)	6.6	330	4400
Shot capacity (oz.)	0.13 - 0.50	8 - 34	413 – 1054
Clamp stroke (in.)	4.33	23.6	133.8
Min. mold thickness (in.)	1.18	7.9	31.5
Platen size (in.)	2.95 x 2.95	40.55 x 40.55	122.0 x 106.3

Table 1.1 Specifications of Injection Molding Machine

Injection molding is employed to provide thin-walled plastic elements for a large type of applications, one in every of the foremost common being plastic housings. Plastic housing may be a thin-walled enclosure, usually requiring several ribs and managers on the inside. These housings area unit employed in a spread of product together with unit appliances, shopper natural philosophy, power tools, and as automotive dashboards. alternative common thin-walled product embrace differing types of open containers, like buckets. Injection molding is additionally accustomed manufacture many everyday things like toothbrushes or tiny plastic toys. several medical devices, together with valves and syringes, area unit factory-made mistreatment injection molding similarly.

1.2 METHOD CYCLE:

The process cycle for injection molding is incredibly short, generally between two seconds and a couple of minutes, and consists of the subsequent four stages:

1. **Clamping -** before the injection of the fabric into the mildew, the 2 halves of the mildew should initial be firmly closed by the clamping unit. every half the mildew is hooked up to the injection molding machine and one 0.5 is allowed to slip. The hydraulicly steam-powered clamping unit pushes the mildew halves along and exerts ample force to stay the mildew firmly closed whereas the fabric is injected. The time needed to shut and clamp the mildew depends upon the machine - larger machines (those with larger clamping forces) would force longer. this point are often calculable from the dry cycle time of the machine.

2. **Injection -** The raw plastic material, sometimes within the sort of pellets, is fed into the injection molding machine, and advanced towards the mildew by the injection unit. throughout this method, the fabric is melted by heat and pressure. The melted plastic is then injected into the mildew terribly quickly and also the buildup of pressure packs and holds the fabric. the number of fabric that's injected is observed because the shot. The injection time is tough to calculate accurately thanks to the advanced and dynamical flow of the melted plastic into the mildew. However, the injection time are often calculable by the shot volume, injection pressure, and injection power.

3. **Cooling -** The melted plastic that's within the mildew begins to chill as shortly because it makes contact with the inside mildew surfaces. because the plastic cools, it'll solidify into the form of the specified half. However, throughout cooling some shrinkage of the half might occur. The packing of fabric within the injection stage permits further material to flow into the mildew and scale back the number of visible shrinkage. The mildew cannot be opened till the desired cooling time has progress. The cooling time are often calculable from many physics properties of the plastic and also the most wall thickness of the half.

4. **Ejection -** once ample time has passed, the cooled half could also be ejected from the mildew by the ejection system, that is hooked up to the rear half the mildew. once the mildew is opened, a mechanism is employed to push the half out of the mildew. Force should be applied to eject the half as a result of throughout cooling the half shrinks and adheres to the mildew. so as to facilitate the ejection of the half, a mildew unharness agent are often sprayed onto the surfaces of the mildew cavity before injection of the fabric. The time that's needed to open the mildew and eject the half are often calculable from the dry cycle time of the machine and may embrace time for the half to fall freed from the mildew. Once the half is ejected, the mildew are often clamped shut for following shot to be injected.

After the injection molding cycle, some post process is usually needed. throughout cooling, the fabric within the channels of the mildew can solidify hooked up to the half. This excess material, at the side of any flash that has occurred, should be cut from the half, generally by mistreatment cutters. for a few sorts of material, like thermoplastics, the scrap material that results from this trimming are often recycled by being placed into a plastic grinder, additionally referred to as regrind machines or granulators, that regrinds the scrap material into pellets. thanks to some degradation of the fabric properties, the regrind should be mixed with stuff within the correct regrind magnitude relation to be reused within the injection molding method.

1.5 MATERIALS

There area unit many sorts of materials that will be employed in the injection molding method. Most polymers could also be used, together with all thermoplastics, some thermosets, and a few elastomers. once these materials area unit employed in the injection molding method, their raw type is sometimes tiny pellets or a fine powder. Also, colorants could also be other within the method to manage the colour of the ultimate half. the choice of a fabric for making injection wrought elements isn't only primarily based upon the specified characteristics of the ultimate half. whereas every material has totally different properties which will have an effect on the strength and performance of the ultimate half, these properties additionally dictate the parameters employed in process these materials. every material needs a distinct set of process parameters within the injection molding method, together with the injection temperature, injection pressure, mildew temperature, ejection temperature, and cycle time. A comparison of some normally used materials is shown below.

Table 1.2 Details of Materials and their Applications

Material Name	Abbreviation	Trade Names	Description	Applications
Acetal	POM	Celcon, Delrin, Hostaform, Lucel	Strong, Rigid, Excellent Fatigue Resistance, Excellent Creep Resistance, Chemical Resistance, Moisture Resistance, Naturally Opaque White, Low / Medium Cost	Bearings, Cams, Gears, Handles, Plumbing Components, Rollers, Rotors, Slide Guides, Valves
Acrylic	PMMA	Diakon, Oroglas, Lucite, Plexiglas	Rigid, Brittle, Scratch Resistant, Transparent, Optical Clarity, Low / Medium Cost	Display Stands, Knobs, Lenses, Light Housin Panels, Reflectors, Signs, Shelves, Trays
Acrylonitrile Butadiene Styrene	ABS	Cycolac, Magnum, Novodur, Terluran	Strong, Flexible, Low Mold Shrinkage (Tight Tolerances), Chemical Resistance, Electroplating Capability, Naturally Opaque, Low / Medium Cost	Automotive (Consoles, Panels, Trim, Vents), Boxes, Guages, Housings, Inhalors, Toys
Cellulose Acetate	CA	Dexel, Cellidor, Satilithe	Tough, Transparent, High Cost	Handles, Eyeglass Frames
Polyamide 6 (Nylon)	PA6	Akulon, Ultramid, Grilon	High Strength, Fatigue Resistance, Chemical Resistance, Low Creep, Low Friction, Almost Opaque / White, Medium / High Cost.	Bearings, Bushings, Gears, Rollers, Wheels
Polyamide 6 / 6 (Nylon)	PA6 / 6	Kopa, Zytel, Radilon	High Strength, Fatigue Resistance, Chemical Resistance, Low Creep, Low Friction, Almost Opaque / White, Medium / High Cost.	Handles, Levers, Small Housings, Zip Ties
Polyamide 11 + 12 (Nylon)	PA 11 + 12	Rilsan, Grilamid	High Strength, Fatigue Resistance, Chemical Resistance, Low Creep, Low Friction, Almost Opaque to Clear, Very High Cost.	Air Filters, Eyeglass Frames, Safety Masks
Polycarbonate	PC	Calibre Lexan, Makrolon	Very Tough, Temperature Resistance, Dimensional Stability, Transparent, High Cost	Automotive (Panels, Lenses, Consoles), Bottles, Containers, Housings, Light Covers, Reflectors, Safety Helmets and Shields.
Polyester - Thermoplastic	PBT, PET	Celanex, Crastin, Lupox, Rynite, Valox	Rigid, Heat Resistance, Chemical Resistance, Medium / High Cost	Automotive (Filters, Handles, Pumps), Bearing Cams, Electrical Components (Connectors, Sensors), Gears, Housings, Rollers, Switches Valves
Polyether Sulphone	PES	Victrex, Udel	Tough, Very High Chemical Resistance, Clear, Very High Cost	Valves

Table 1.2 Details of Materials and their Applications

II. LITERATURE REVIEW

The following area unit the a number of the articles reviewed associated with the current work.

M. Stanek, D. Manas, M. Manas [1] have determined the acceptable injection pressure, velocity, worth and time of packing pressure, etc. by improvement. Their article deals with description of mildew flow Plastics Xpert (MPX) system and its usage in improvement of injection molding method on real half throughout its production. Alireza Akbarzadeh and Mohammad Sadeghi [2] , investigated impact of injection molding parameters on the shrinkage in plastic (PP) and cinnamene (PS). the connection between input and output of methodology} is studied mistreatment regression method and Analysis of Variance (ANOVA) technique. To do this, existing knowledge is employed. the chosen input parameters area unit melting temperature, injection pressure, packing pressure and packing time. impact of those parameters on the shrinkage of on top of mentioned materials is studied mistreatment mathematical modeling. For modeling the method, differing types of regression equations together with linear polynomial, multinomial and index operate, area unit accustomed interpolate experiment knowledge. Next, mistreatment step backward elimination and ninety fifth confidence level (CL), insignificant parameters area unit eliminated from model. to envision validity of the PP model, parametric statistic of every model is calculated and also the best model is chosen. an equivalent procedure is recurrent for the note model. Finally, optimum levels of the input parameters that minimize shrinkage, for each materials area unit determined. Invasive Weed improvement (IWO) algorithmic rule is applied on the developed mathematical models. The improvement results show that the projected models and algorithmic rule area unit effective in resolution the mentioned issues.

Mohd.rizwan Hamsin, Azuddin Mamat and Aznijar Ahmad-Yazid[3], This paper describes the look and analysis of plastic injection mildew reconciliation runner. The runners were designed supported Ellis Model, a viscousness model of flow network consisting of parts and nodes. 4-coefficients viscousness model and temperature dependence Ellis Model were employed in order to cut back the number of process analysis by FEA software system. A Cross WLF viscousness model was employed in the FEA analysis. FEA simulation of injection molding was conducted for eight and sixteen cavity runners. Runner layout was assumed as pressure at the top of every component acting as associate initial and final condition. The length and size of the runner are often adjusted to suit the condition that had been chosen. Flow rates at every component resulting in gates were set to simulate the specified pressure drop. the ultimate condition for the primary component was set because the initial condition for following component. By using Ellis model, it had been shown that the calculated results area unit just like the result obtained through simulation. The model utilized has with success shown associate equal filling time for every cavity, associate equal pressure at every gate similarly as uniform half filling. A prophetical FEA performed before actual producing is useful so as to provide sensible molds. S. H. Park1, W. I. Lee1, S. N. Moon1, Y.-E. Yoo2, Y. H. Cho3[4] ,Poor filling happens throughout the injection mildewing method of micro- or nano-scale patterns primarily as a result of the recent compound soften speedily cools and its skin quickly solidifies upon contact with the mold surface. during this study, it's projected to use synthetic resin terephthalate (PET) film coated with brindled ployurethan salt (PUA) as an efficient bounds. It will considerably hinder heat transfer into the mildew throughout the molding method and so might keep the soften viscousness low for extended period. As a result, the replication would be improved not solely throughout the filling part however additionally throughout the packing part. so as to verify the validity of the utilization of compound stamper the melt-film interface temperature was evaluated by numerical simulation. Experimental results indicated that patterns possessing widths inside the vary of 1 to tens of micrometers and a height of roughly 10!m were with success stuffed and Diamond State wrought.

Wei Guo,*Lin Al Faran Huajie Mao and Zhenghua Meng[5], In terms of injection process parameters, a mathematical model for prediction of warpage was developed supported style of experiments (DOE). First, the 5 most authoritative parameters were screened by mistreatment fragmental factorial style (FFD): soften temperature, fluid temperature, injection time, V/P exchange and mildew temperature. Second, considering the opposite four principal process parameters except the soften temperature, the predicting mathematical model was supported by mistreatment central composite style (CCD) of experiments and metallic element simulation. Finally, the results of applied mathematics analysis were collected from software system mildew flow. The results prompt that the mathematical model are often accustomed predict warpage with adequate accuracy. Hence, it indicated that corrective and repetitive style steps are often initiated and enforced for higher quality of product while not resorting to physical trials in plastics injection mildew by mistreatment this predicting mathematical model. M. R. Mani, R. Surace, P. Ferreira, J. Segal, I. Fassi and S. Ratchev [6], Micro-injection moulding is turning into more and more vital among the offered processes for production of micro-electromechanical systems (MEMS) and Microsystems technologies (MSTs), and better range of applications of this method, like medical and part applications, achieving top quality elements with compound product is being factory-made by this method. thanks to the sensitive nature of high dimensional accuracy is crucial. during this work, a style of experiment (DoE) approach is employed. The aim is to check the results of 3 method parameters that area unit normally used for analysis during this domain, on the dimensional accuracy of small channels with totally different sizes; they're injection rate, injection pressure, and soften temperature. The study focuses on 2 polymers, poly oxy methylene radical (POM) and liquid compound (LCP). Experimental results showed that higher soften temperature and injection pressure resulted in higher dimensional accuracy. all the same, high settings for the 3 parameters resulted in higher share of flash in most cases. lastly, the foremost authoritative factors were shown to be soften temperature and injection pressure.

III. INJECTION MOLDING METHOD CONDITIONS

Injection molding is associate engineering technology, it's associated with the contents of the plastic into helpful and might maintain the first performance of the product. The importance of injection molding process conditions have an effect on the plastics flow and cooling the temperature, pressure and also the corresponding time for every role.

3.1.1 TEMPERATURE MANAGEMENT

i. Barrel Temperature: Injection molding method needs temperature management of barrel temperature, nozzle temperature and die temperature. the primary 2 temperatures primarily affected the plastics and plastic flow, the latter primarily affects the temperature of plastic flow and cooling. every contains a totally different plastic flow temperature, an equivalent kind of plastic, as a result of totally different sources or brands, the flow temperature and decomposition temperature may be a distinction, this is often thanks to {different|totally totally different|completely different} relative molecular mass and relative molecular mass distribution caused by differing types of plastic injection machine plasticizing method is different, therefore choose the barrel temperature isn't an equivalent.

ii. Nozzle Temperature: Nozzle temperature is sometimes slightly not up to the best temperature of the barrel, this is often to forestall soften through nozzles might occur within the "drooling development." Nozzle temperature, not too low, otherwise it'll end in the first soften natural process and also the nozzle block * or early atmospheric phenomenon as feed into the mildew cavity to have an effect on the performance of product

iii. mildew Temperature: mildew temperature on the product's inherent quality of performance and apparent nice impact. mildew temperature is set by the provision of the plastic crystal, the scale and structure of product, performance necessities, and alternative process conditions (melt temperature, injection speed and injection pressure, molding cycle, etc.).

3.1.2 PRESSURE MANAGEMENT

The process of pressure injection pressure and injection pressure of 2 forms of plastics and plastic directly have an effect on the standard of plastics and product.

i. Plasticizing Pressure: (back pressure) mistreatment screw-type injection machine, the screw high of the soften within the screw by turning the pressure back once referred to as plasticization pressure, additionally called back pressure. the scale of this pressure is thru the mechanism valve to be adjusted. The injection, the plasticization pressure, the scale of screw speed with all an equivalent, a rise of pressure which will increase the plasticizing temperature of the soften, however can scale back the speed of softener. additionally, the redoubled pressure usually build plastics soften temperature uniformity, pigment mixed equally and soften within the gas discharge. traditional operation, the plastics pressure to choose in making certain sensible quality product underneath the premise of the lower the higher, its specific worth is employed in plastics with totally different varieties, however rarely over twenty kilogram / sq. cm.

ii. Injection Pressure: within the current production, most of the injection machine injection pressure is to screw the highest on the plastic plunger or by the pressure (the pressure from the oil line conversion to a) the yardstick. Injection pressure in injection molding is that the role vie by overcoming the flow of plastic from the barrel cavity flow resistance, given the speed of soften filling and compaction of the soften for.

3.1.3 MOLDING CYCLE

Complete a course of your time needed for injection molding, aforesaid molding cycle, additionally called the molding cycle. It really includes the subsequent components: forming cycle: cycle time directly have an effect on labor productivity and instrumentality utilization. Therefore, within the production method ought to be the premise of guaranteeing quality, shorten the molding cycle, all the time. Throughout the molding cycle time for injection and cooling time is most significant, the standard of their product have a decisive impact. Injection time of the filling time is directly reciprocally proportional to the filling rate, the assembly of filling usually regarding 3-5 seconds.

Injection time of the holding pressure time is plastic on the cavity pressure time, the injection time within the proportion of enormous, typically regarding 20-120 seconds. Frozen at the gate to soften before the holding time of what quantity impact on the accuracy of product size, if within the future, wasn't affected. Holding time is additionally the foremost favorable worth, familiar it depends on the soften temperature, mildew temperature and also the main channel and also the gate size. Cooling time depends primarily on the thickness of product, plastic thermal properties and crystal properties, and mildew temperature, etc.. the top of the cooling time ought to be to make sure that once the merchandise doesn't cause changes in ejection principle, the temporal order of the final cooling of regarding between thirty to a hundred and twenty seconds, cooling time is just too long not necessary, not solely reduces the potency of production of advanced work items can difficulties caused by husking, forced ejection time and maybe even husking the strain. alternative molding cycle time was whether or not the method and production automation, and continuous and continuous, and also the degree of automation and then on.

Automatic injection molding cycle is sustained within the same order, an equivalent issue over and yet again. Cycle has 3 main parts:

Mold Time Filling Time Mold Closure Time;

3.2 FACTORS POIGNANT MILDEW

the primary thought is that the speed and movement distance of mildew, the mildew opens and also the high of the work piece within the method of moving the space of movement ought to be reduced therefore as to not waste time, of course, die mobile should be ample for the work piece before the closure once more rid mildew, therefore, necessary to maneuver the work piece ejection shorter distance, then it's among the less spent, once the injection molding machine in condition, from high to low to open the highest of the reborn are often terribly sleek. instrumentality needs some maintenance to complete these changes in speed, however they will be wrought to pay less time, save time and find a come again and again. what is more, the acceptable periodic maintenance to make sure that the retardation are often recurrent whenever. manufacture the mildew clamping pressure of your time is another delay in gap hours, this point through mechanical wear and potential failure of hydraulic valves, therefore periodic maintenance to keep up sensible mechanical operative condition.

3.4 ISSUES SWEET-FACED BY THE CORPORATE THROUGHOUT INJECTION MOULDING METHOD

In most of the injection molding production unit’s, throughout production downside is sweet-faced once filling the fabric within the cavity space. The makers modification the process parameters by trial and error technique. By trial and error technique, there's ton of wastage of fabric, time, and power. At an equivalent time value of the part will be redoubled. In this project on top of downside is corrected by taking software system support of Plastic authority. mistreatment this software system, the standard of filling of fabric by dynamical process parameters before planning to producing are often analyzed.

Mould flow analysis is additionally done by taking “One Gate Location, 2 gate locations and 3 gate locations”.

IV. INTRODUCTION TO GO LAMPS

A light may be a lamp hooked up to the front of a vehicle to lightweight the road ahead. light performance has steady improved throughout the auto age, spurred by the good inequality between daytime and nighttime traffic fatalities: the U.S. National route Traffic Safety Administration states that almost half all traffic-related fatalities occur within the dark, despite solely twenty fifth of traffic travel throughout darkness. While it's common for the term light to be used interchangeably in informal discussion, whereas light properly refers to the beam of sunshine made and distributed by the device. Other vehicles, like trains and craft, area unit needed to own headlamps. Bicycle headlamps area unit usually used on bicycles, and area unit needed in some jurisdictions. they will be steam-powered by electric battery or alittle electrical generator on the wheel.

Additionally automotive scotopic vision systems work to supplement headlamps.

4.2 MODELS OF HEAD LAMP WITH TOTALLY DIFFERENT GATE LOCATIONS

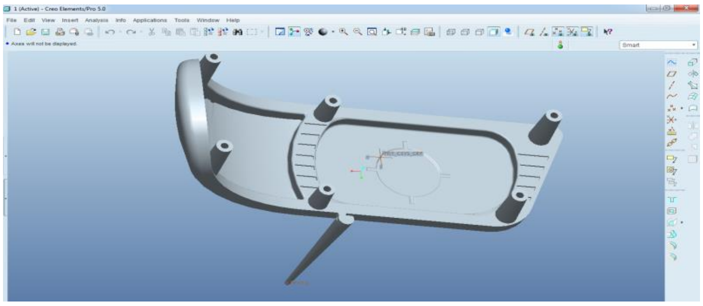

Fig. 4.1 Single Gate

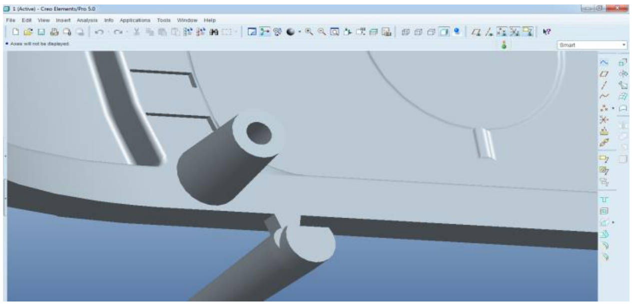

Fig. 4.2 Gates

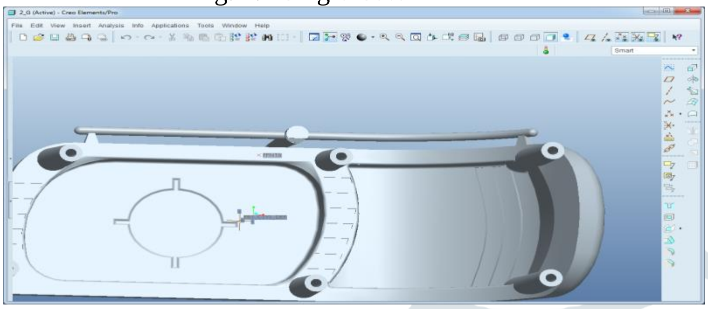

Fig. 4.3 Gate

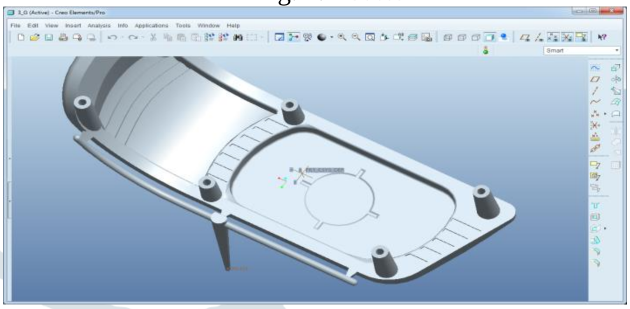

Fig. 4.4 Gates

4.3 MOULD FLOW ANALYSIS

Mould flow, 3D solids-based plastics flow simulation that enables plastics half styleers to see the manufacturability of their elements throughout the preliminary design stages and avoid potential downstream issues, which may result in delays and price overruns. Following area unit the benefits:

- Optimize the half wall thickness to attain uniform filling patterns, minimum cycle time and lowest half price establish and eliminate cosmetic problems like sink marks, weld lines and air traps.
- Determine the most effective injection locations for a given half style.

Mould flow analysis offers you the power to keep up the integrity of your product styles. It provides you the tools to quickly optimize half styles and check the impact of essential style selections on the manufacturability and quality of the merchandise early within the style method.

4.4 PLASTC AUTHORITY

Plastic authority is associate add on analysis package for Pro/Engineer, specially for plastic injection moulding. For doing this analysis, once drawing the desired object choose applications □ plastic advisor □ decide {datum|data purpose|information} point for injection location □ ok.

The icons offered area unit

1) Moulding Parameter Icon
2) Specify Injection Location
3) Special Analysis Icon
4) Start Analysis Icon.

When we click on the moulding parameter icon a separate window opens and there {we will|we will|we are able to} choose {the needd|the specified|the desired} materials and that we can specify the require injection conditions like mould temperature, injection pressure.

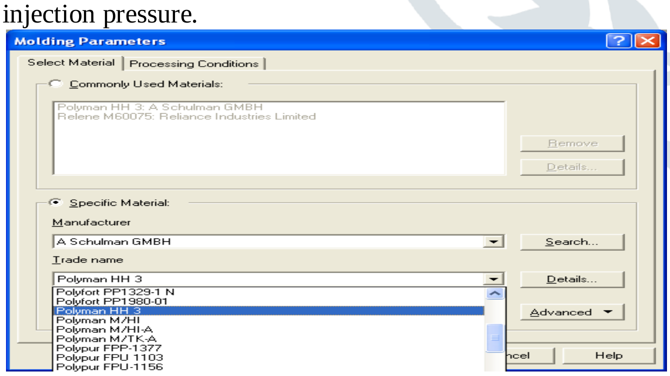

Fig. 4.6 Molding Parameters

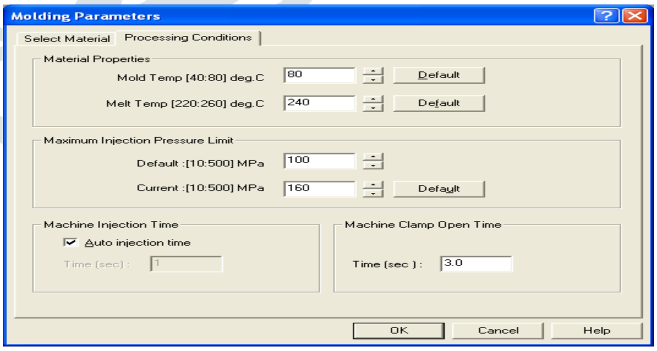

Fig. 4.7 process Conditions

After specifying the equipage and process conditions, click on the specify injection location icon and specify the desired location purpose. Then click on the run analysis icon and choose plastic flow analysis and click on on begin button.

4.5 MATERIAL USED FOR HEAD LIGHTWEIGHT

Acrylonitrile/butadiene/styrene (ABS) Primarily used for the manufacture of housings, covers and linings, that includes the subsequent properties:

- excellent properties of toughness, strength and rigidity;
- they're opaque, attain high surface polish i. e. have well polished surface;
- sensible chemical resistance and resistance to temperature of 80-105°C;
- modulus in tension is from 1500-2700 MPa, and once fiber glass strengthened even up to 5500 MPa.

4.6 MOULD FLOW ANALYSIS - SINGLE GATE LOCATION at 130 MPa Pressure

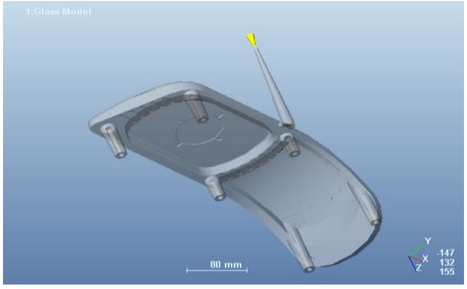
Fig. 4.8 Glass Model

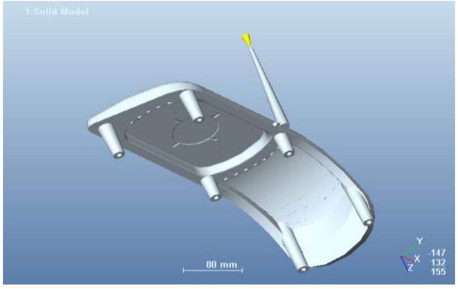
Fig. 4.9 Solid Mode

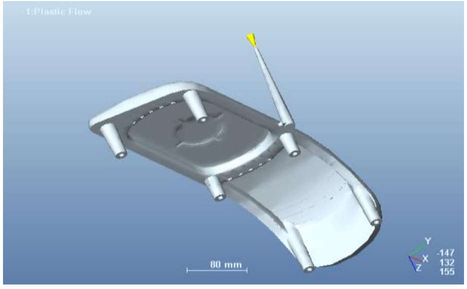
Fig. 4.10Plastic Flow

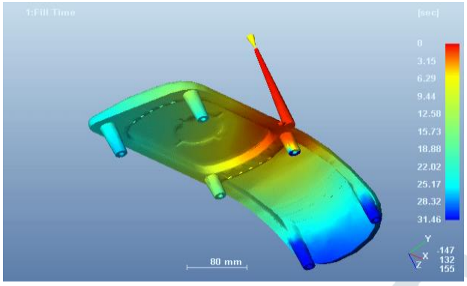
Fig. 4.11Fill Time

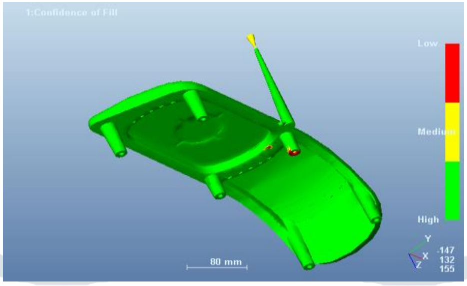
Fig. 4.12Confidence of Fill

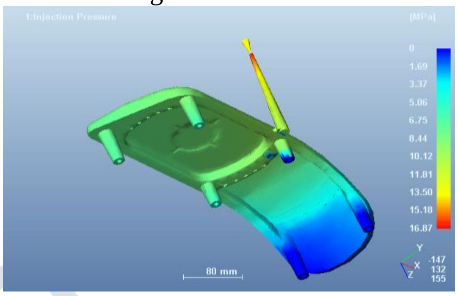
Fig. 4.13 Injection Pressure

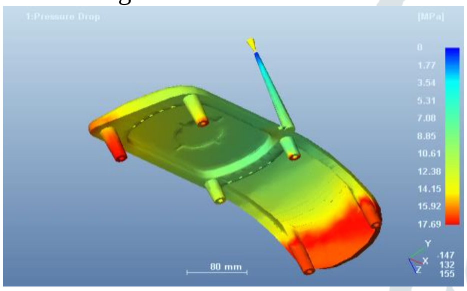
Fig. 4.14 Pressure Drop

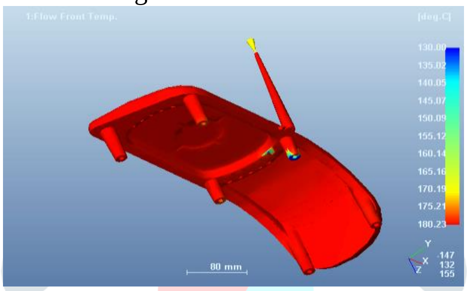
Fig. 4.15 Flow Front worker

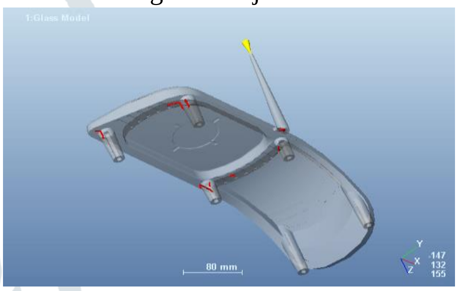
Fig. 4.16 Weld Lines

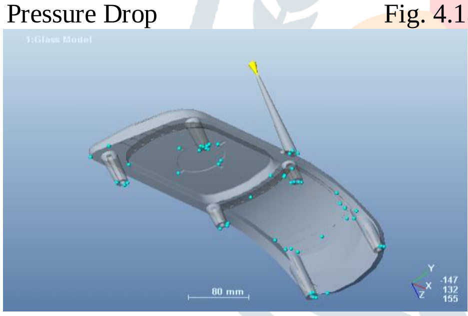
Fig. 4.17 Air Traps

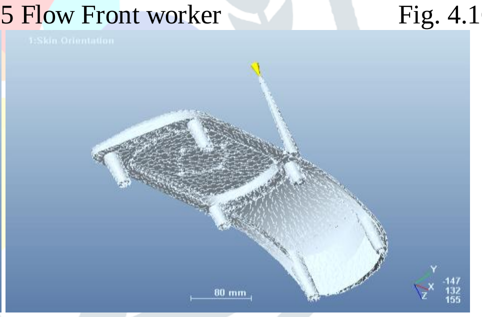
Fig. 4.18 Skin Orientation

4.6.2 CASE 2 – BY INCREAING INJECTION PESSURE SUMMARY: at 180 MPa Pressure

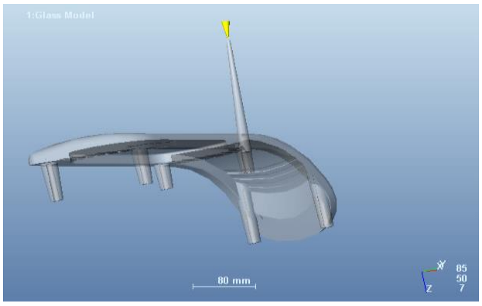
Fig. 4.19 Glass Model

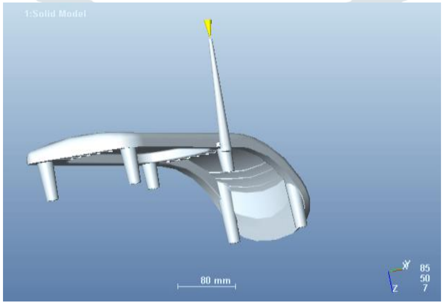
Fig. 4.20 Solid Mode

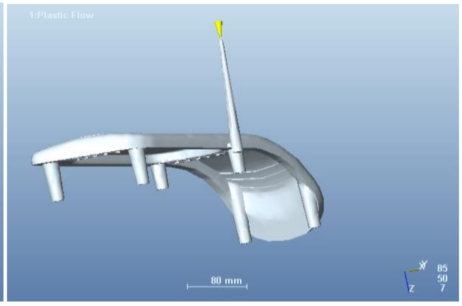
Fig. 4.21Plastic Flow

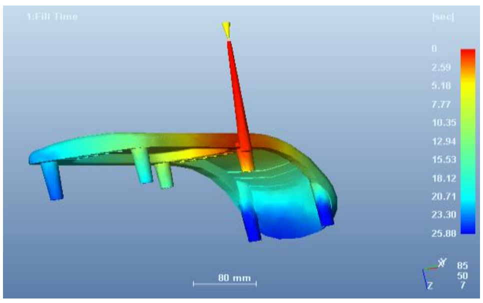
Fig. 4.22 Fill Time

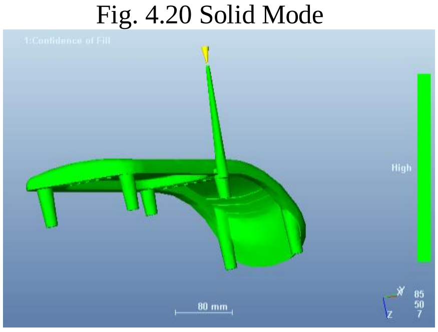
Fig. 4.23 Confidence Of Fill

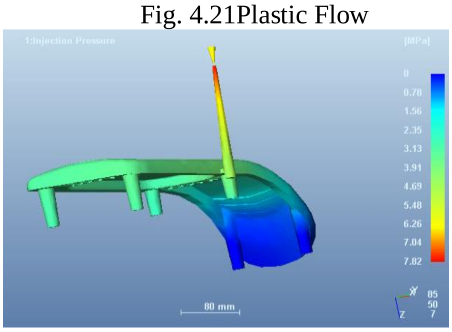
Fig. 4.24 Injection Pressure

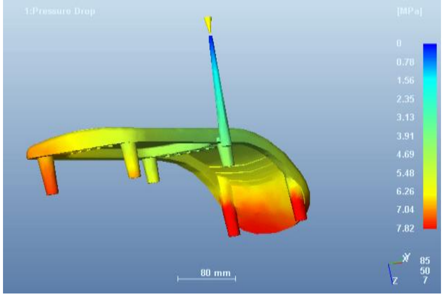
Fig. 4.25 Pressure Drop

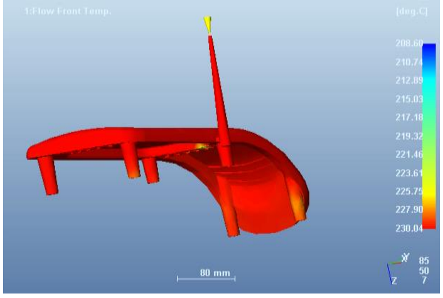
Fig. 4.26 Flow Front worker

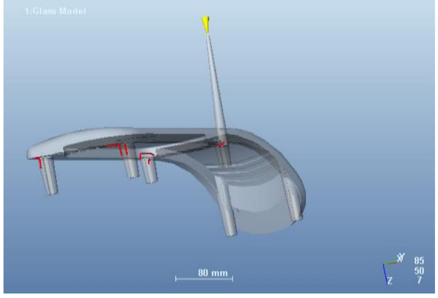
Fig. 4.27 Weld Lines

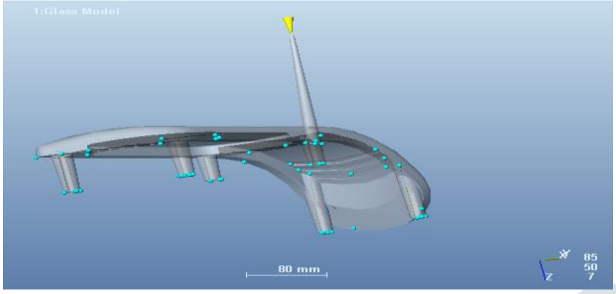
Fig. 4.28 Air Traps

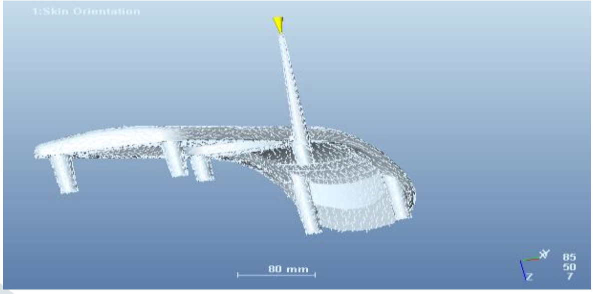
Fig. 4.29 Skin Orientation

4.7 MOULD FLOW ANALYSIS – 2 GATE LOCATIONS

4.7.1 CASE 1 – AT LOW INJECTION PRESSURE at 130 MPa Pressure

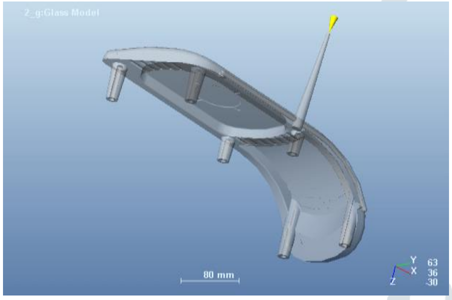
Fig. 4.30 2 Glass Model

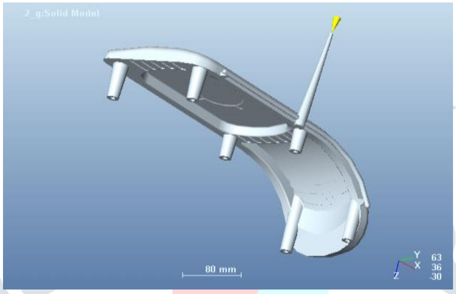
Fig. 4.31 2 Solid Model

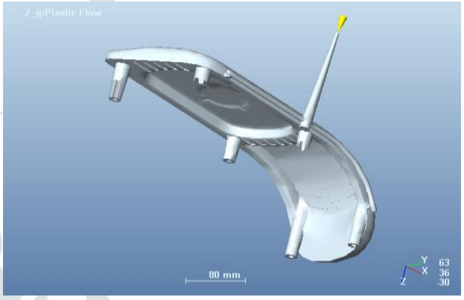
Fig. 4.32 2 Plastic Flow

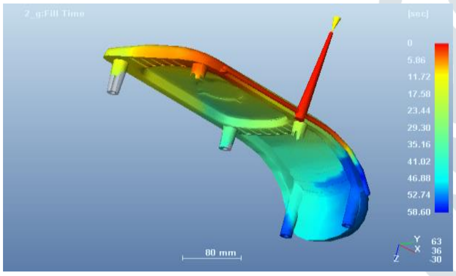
Fig. 4.33 Fill Time

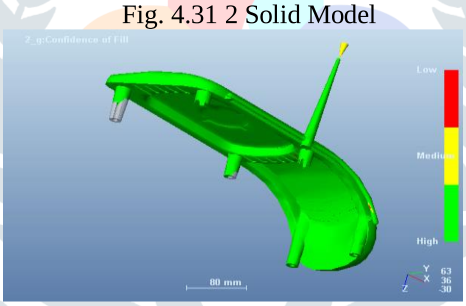
Fig. 4.34 Confidence Of Fill

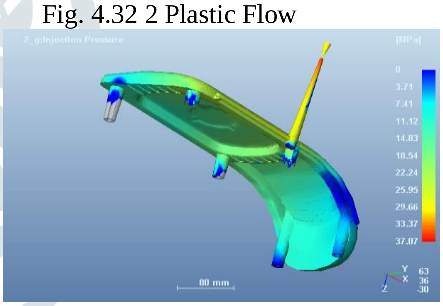
Fig. 4.35Injection Pressure

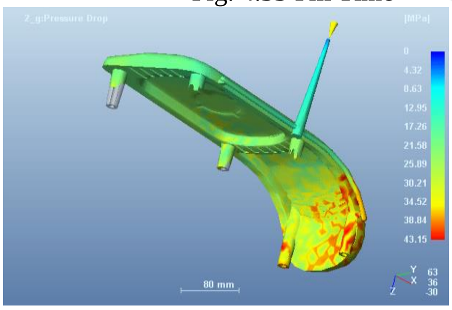
Fig. 4.36Pressure Drop

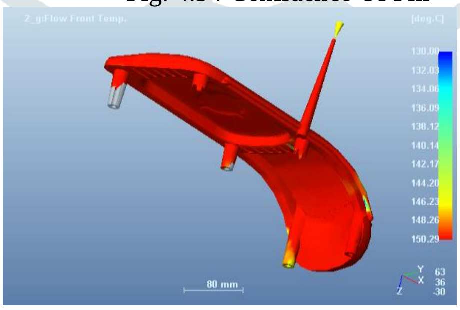
Fig. 4.37 Flow Front worker

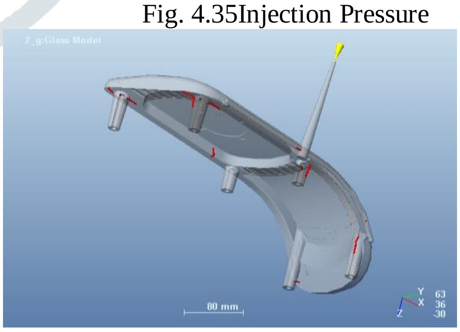
Fig. 4.38 Weld Lines

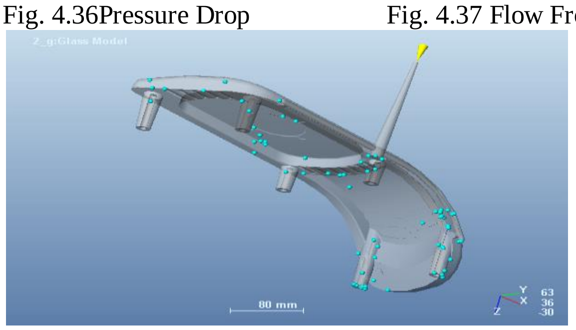
Fig. 4.39 Air Traps

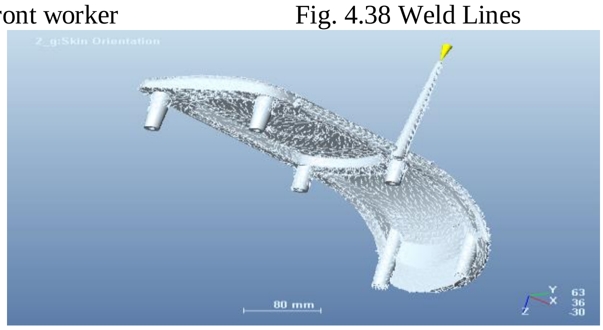
Fig. 4.40 Skin Orientation

7.2 CASE 2 – BY INCREAING INJECTION PRESSURE at 180 MPa Pressure

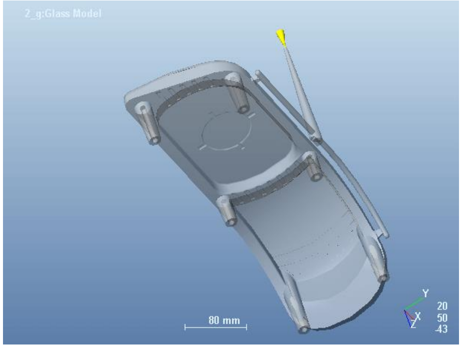
Fig. 4.41Glass Model

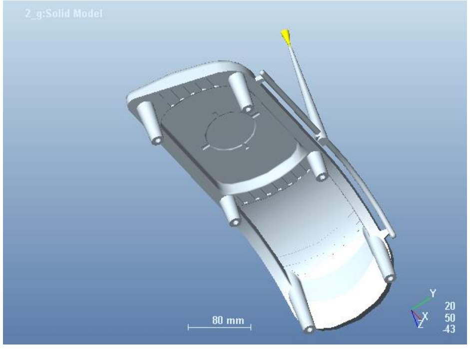
Fig. 4.42 Solid Mode

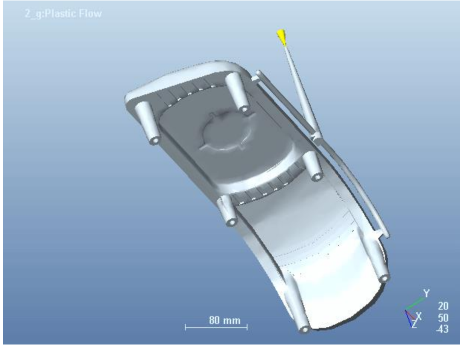
Fig. 4.43Plastic Flow

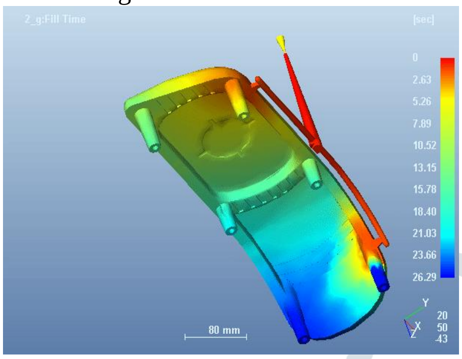
Fig. 4.44 Fill Time

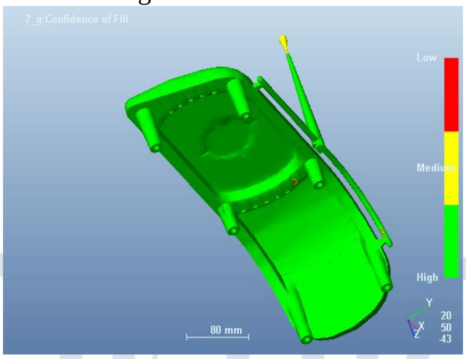
Fig. 4.45Confidence Of Fill

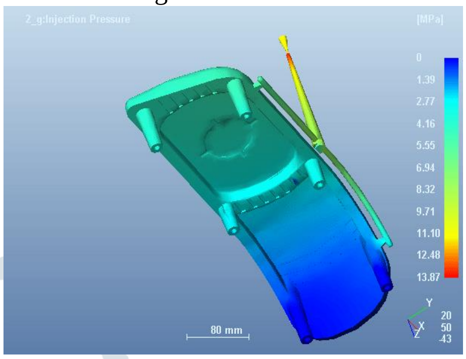
Fig. 4.46 Injection Pressure

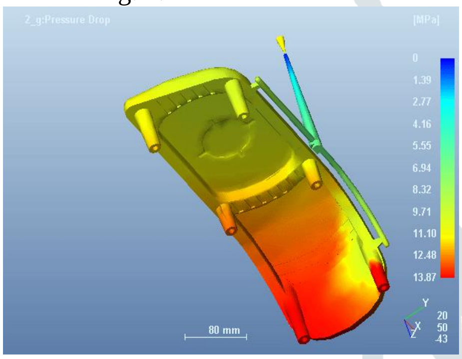
Fig. 4.47 Pressure Drop

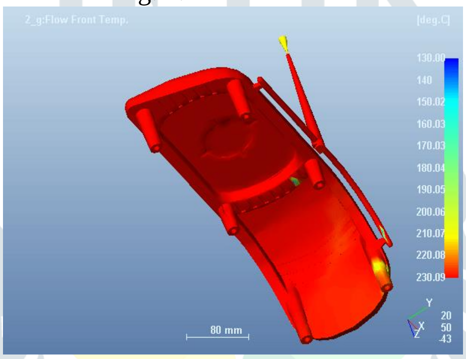
Fig. 4.48Flow Front Temp

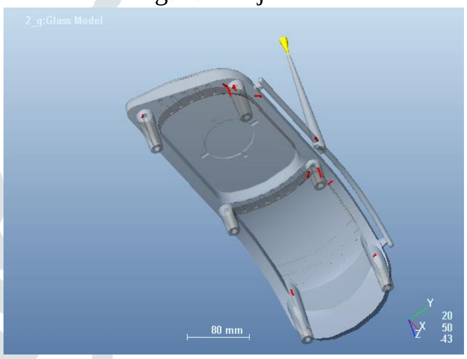
Fig. 4.49 Weld Lines

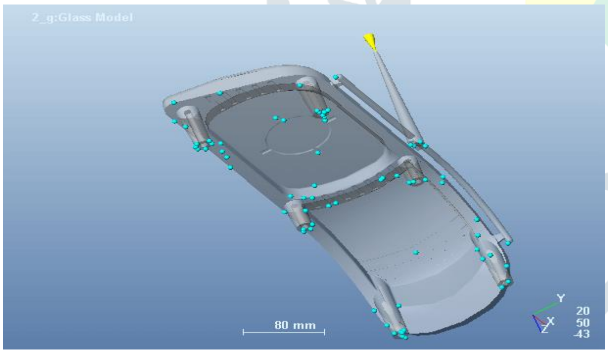
Fig. 4.50Air Traps

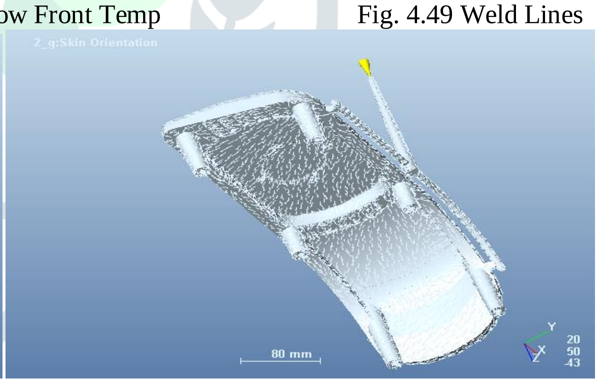
Fig. 4.51Skin Orientation

4.8.MOULD FLOW ANALYSIS - Three Gate Locations

4.8.1 CASE 1 – AT LOW PRESSURE at 130 Mpa Pressure

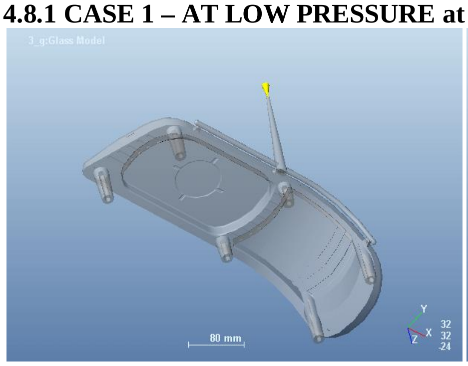
Fig. 4.52 Glass Model

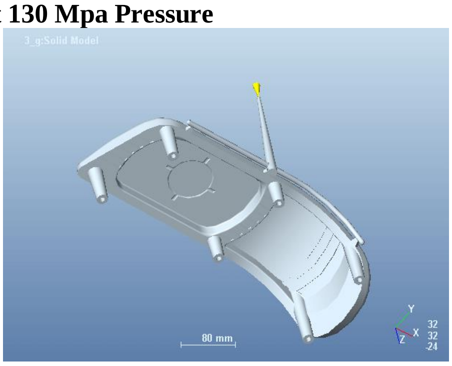
Fig. 4.53Solid Mode

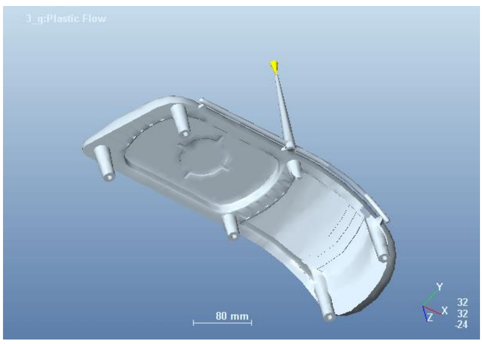
Fig. 4.54 Plastic Flow

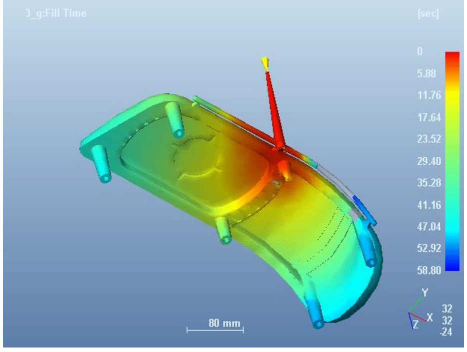

Fig. 4.55 Fill Time

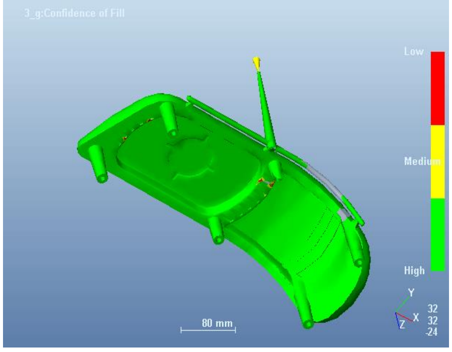

Fig. 4.56Confidence Of Fill

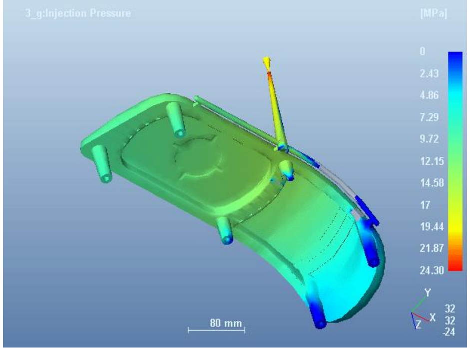

Fig. 4.57 Injection Pressure

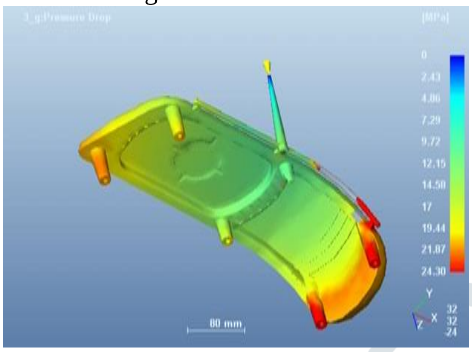

Fig. 4.58Pressure Drop

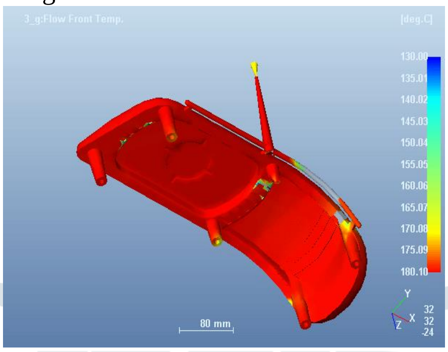

Fig. 4.59 Flow Front Temp

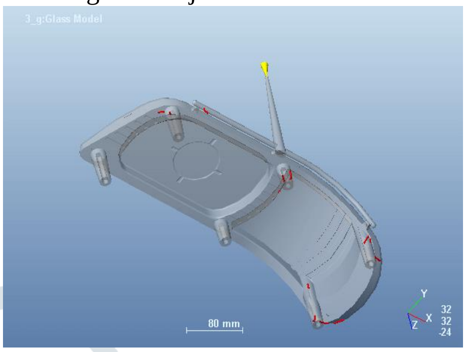

Fig. 4.60 Weld Lines

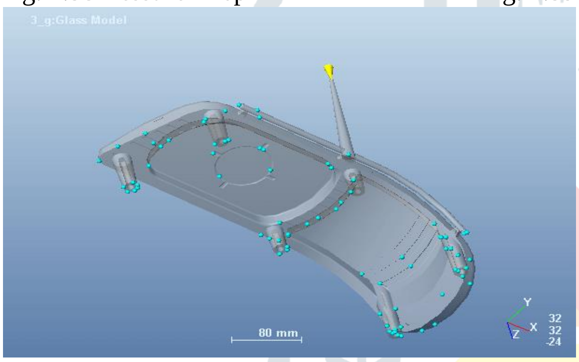

Fig. 4.61 Air Traps

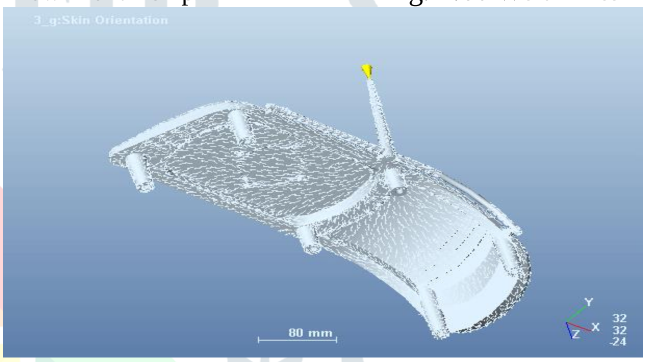

Fig. 4.62 Skin Orientation

4.8.2 CASE 2 – BY INCREASING INJECTION PRESSURE at 180 Mpa Pressure

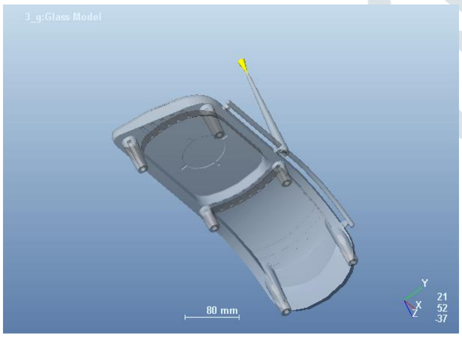

Fig. 4.63Glass Model

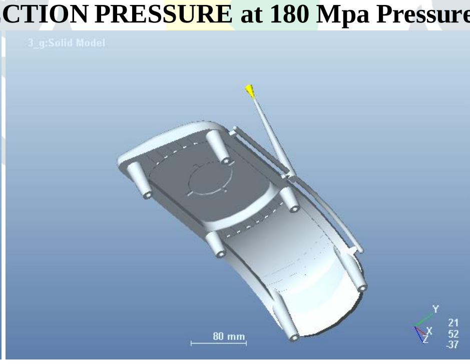

Fig. 4.64Solid Mode

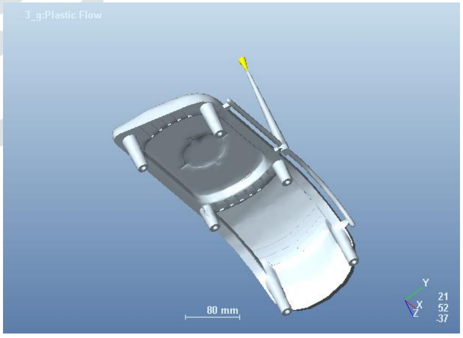

Fig. 4.65Plastic Flow

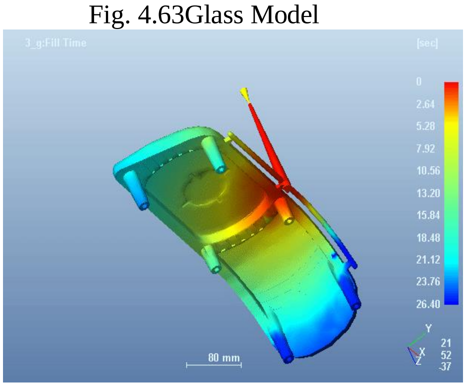

Fig. 4.66 Fill Time

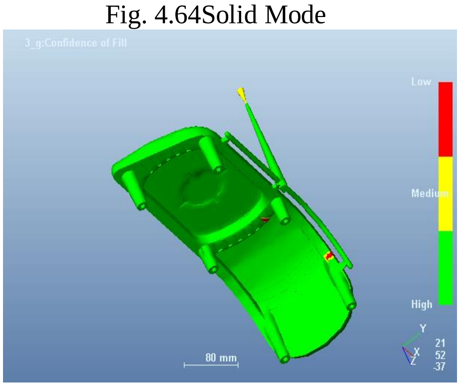

Fig. 4.67 Confidence Of Fill

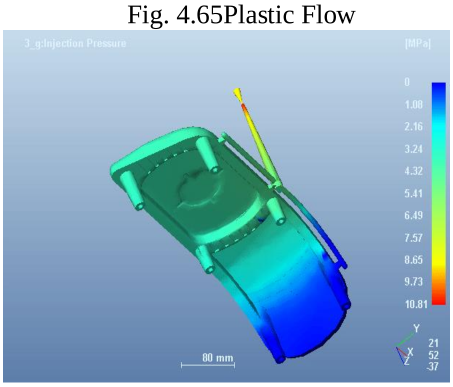

Fig. 4.68 Injection Pressure

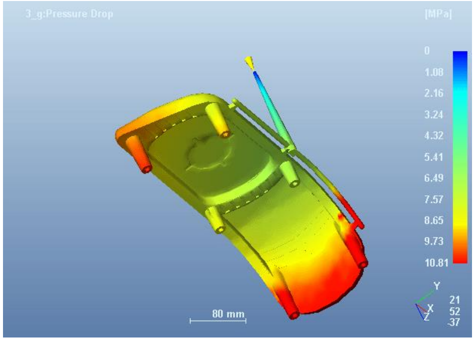

Fig. 4.69Pressure Drop

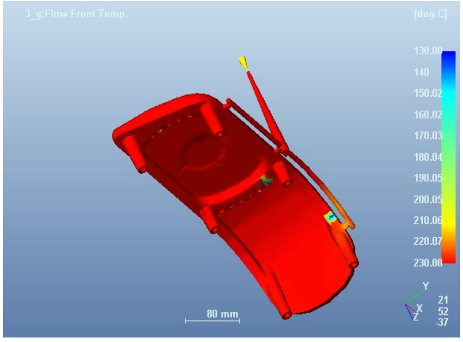

Fig. 4.70 Flow Front Temp

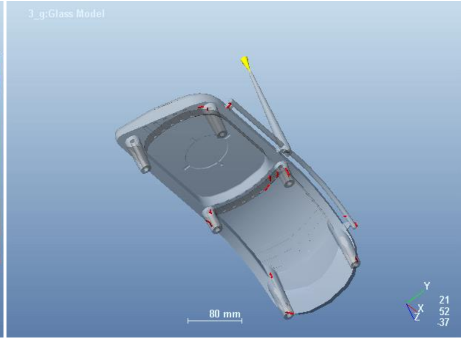

Fig. 4.71Weld Lines

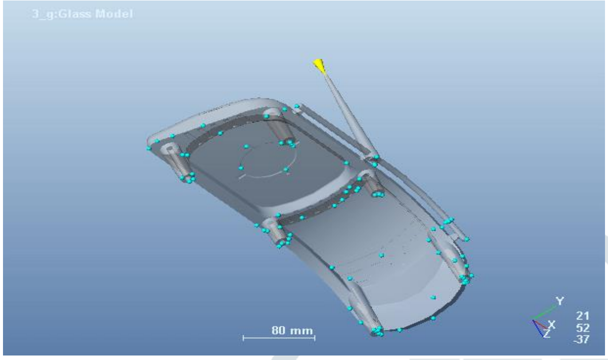

Fig. 4.72 Air Traps

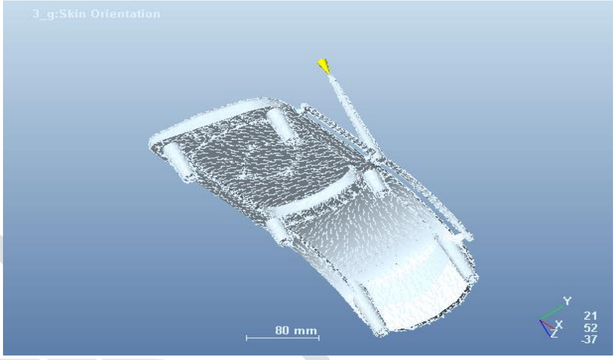

Fig. 4.73Skin Orientation

RESULTS AND DISCUSSIONS

During this gift work whole Six (6) variations of Head Lamp Model of Alto automobile area unit analyzed by mistreatment plastic authority professional / engineer software system (Injection Moulding). the subsequent table.5.1 shows analysis and ends up in following parameters like, Filling Time, Confidence Filling Level, Injection Pressure, Flow Front Temperature and Pressure Drop.

S.No.	Model with Number of Gate Locations	Input Pressure in MPa	Fill Time in Seconds	Confidence of Fill in mm	Injection Pressure in MPa	Pressure Drop in MPa	Flow Front Temperature in °C
1	Single Gate Location	130	31.46	155	16.87	17.69	180.23
2	Two Gate Location	130	58.60	63	37.07	43.15	150.29
3	Three Gate Location	130	58.80	32	24.30	24.30	180.10
4	Single Gate Location	180	25.88	85	7.82	7.82	230.04
5	Two Gate Location	180	26.29	50	13.87	13.87	230.09
6	Three Gate Location	180	26.40	52	10.81	10.81	230.08

Table 5.1 Details of various Gate Locations with their Results Develop

CONCLUSIONS

During this project work, the injection mould flow method is globalized by variable range of gates from One, 2 . The input pressure is varied from a hundred thirty MPa to one hundred eighty MPa and also the analysis is dole out to see the optimized Injection Mould flow, Input Pressure at the side of range of Gate Locations.

From the analysis, it's ascertained that, associate optimized flow is ascertained for a 2 gate location model at a hundred thirty MPa input pressure. during this case, confidence levels area unit ascertained to be high with a price of injection pressure thirty seven.07 MPa, Flow Front Temperature of 230.09 MPa, Pressure Drop forty three.15.

It's terminated that a 2 gate locations at {180|one hundred eighty|a hundred associated eighty} MPa Input pressure will manufacture an optimized heat temperature cowl in comparison with remaining cases.

In the project work, the optimum range of gates needed to fill the part head lightweight with least defects is analyzed. the quantity of gates taken united, 2 and 3. The injection pressure given by the corporate at this time is 130MPa; analysis is finished by increasing the pressure to 180MPa.

By perceptive the analysis results, for all models of gates, the injection time is a smaller amount once injection pressure is redoubled than the first injection pressure. the arrogance of fill is high once One Gate Location is taken however alternative defects like flow marks, weld lines are a lot of. the arrogance of fill once 2 gates is taken is slightly but that of 1 Gate Locations however the opposite defects are often reduced. By taking 3 gate locations, the wastage of fabric are a lot of.

So it's terminated that mistreatment 2 gate locations and by increasing the injection pressure offers higher filling results.

REFERENCES

1. M. Stanek, D. Manas, M. Manas and O. Suba "Optimization of Injection Molding Process" International Journal Of Mathematics And Computers In Simulation Issue 5, Volume 5, 2011, PP413-421.
2. Alireza Akbarzadeh and Mohammad Sadeghi "Parameter Study in Plastic Injection Molding Process using Statistical Methods and IWO Algorithm" International Journal of Modeling and Optimization, Vol. 1, No. 2, June 2011.
3. Mohd.rizwan Hamsin, Azuddin Mamat and Aznijar Ahmad-Yazid" Design and analysis Of Multi-Cavity Traditional and H-Branching Runners For plastic injection Mold" International journal of Design and Manufacture.
4. S. H. Park, W. I. Lee, S. N. Moon, Y.E. Yoo, Y. H. Cho "Injection molding micro patterns with high aspect ratio using a polymeric flexible stamper" EXPRESS Polymer Letters Vol.5, No.11 (2011) 950–958 DOI: 10.3144/expresspolymlett.2011.93.
5. Wei Guo,*Lin Hua Huajie Mao and Zhenghua Meng" Prediction of warpage in plastic injection molding based on design of experiments "Journal of Mechanical Science and Technology 26 (4) (2012) 1133~1139 DOI10.1007 /s12206-012-0214-0.
6. R. Mani, R. Surace, P. Ferreira, J. Segal, I. Fassi and S. Ratchev **"**Process Parameter Effects on Dimensional Accuracy of Micro-Injection Moulded Part" J. Micro Nano-Manuf. 1(3), 031003 (Aug 12, 2013) (8 pages) doi:10.1115/1.4025073.
7. Wen-Chin Chen Gong-Loung Fu Pei-Hao TaiWei-Jaw Deng" Process parameter optimization for MIMO plastic injection molding via soft computing "Graduate School of Industrial Engineering and System Management, Chung Hua University No. 707, Sec. 2, Wu Fu Rd., Hsinchu 30012, Taiwan Graduate Institute of Technology Management, Chung Hua UniversityNo.707,Sec.2,WuFuRd.,Hsinchu30012,Taiwan.
8. P.K. Bharti "Recent methods for optimization of plastic injection molding process a retrospective and literature review" International Journal of Engineering Science and Technology Vol. 2(9), 2010, 4540-4554.
9. Gantar, G. – Glojek, A. – Mori, M. – Nardin, B. – Sekavčnik, M.Gašper Gantar - Andrej Glojek Mitja Mori," Resource Efficient Injection Moulding with Low Environmental Impacts" Strojniški vestnik - Journal of Mechanical Engineering 59(2013)3, 193-200 DOI:10.5545/sv-jme.2012.661.
10. T. Stachowiak*, T. Jaruga[10] "Structure of gas-assisted injection moulded parts" Department of Polymer Processing and Production Management, Czestochowa University of Technology, Al. Armii Krajowej 19c, 42-200 Częstochowa, Poland.
11. Sharifah Imihezri Syed Shaharuddin, Mohd. Sapuan Salit, Edi Syams Zainudin "A Review of The Eect of Moulding Parameters on the performance of Polymeric Composite Injection Moulding", Turkish J. Eng. Env. Sci.30 (2006), University Putra Malaysia, Department of Mechanical and Manufacturing Engineering 43400 Serdang, Selangor-MALAYSIA.
12. E. Boci¹Ga, T. Jaruga*, K. Lubczyñska, A. Gnatowski "Warpage of injection moulded parts as the result of mould temperature difference" Department of Polymer Processing and Production Management, Czestochowa University of Technology, Al. Armii Krajowej 19c, 42-200 Czêstochowa.
13. Shaik Mohamed, Mohamed Yusoff Jafri Mohd. Rohani Wan Harun Wan Hamid& Edly Ramly" A Plastic Injection Molding Process Characterization Using Experimental Design Technique" Formal technology, 41(A) Dis. 2004: 1–16© University Technologies Malaysia.
14. Alexandre Thiriez and Timothy Gutowski "An Environmental Analysis of Injection Molding" Department of Mechanical Engineering Massachusetts Institute of Technology Cambridge, MA, USA.
15. Wong, C.T., Shamsuddin Sulaiman, Napsiah Ismail & A.M.S. Hamouda **"**Design and Simulation of Plastic Injection Moulding Process", Department of Mechanical and Manufacturing Engineering University Putra ISSN:0128-7680 ,43400UPM,Serdang,Selangor,Malaysia.
16. OYETUNJI, A" Development of small injection moulding machine for forming small plastic articles for small-scale industries" Journal of Engineering Science and Technology Vol. 5, No. 1 (2010) 17 - 29 © School of Engineering, Taylor's University College.