International Journal of Trend in Research and Development, Volume 3(3), ISSN: 2394-9333
www.ijtrd.com

Design & Development of Plastic Injection Mold for Plastic Clip of School Bag

[1]Harshvardhan Patil, [2]Nikhil Deshpande, [3]Sudarshan Bhambore, [4]Kuldeepak Lokhande,
[5]Aditya Kurlapkar and [6]Pritam Patil, [1]Assistant Professor, [2,3,4,5,6]Students,
[1,2,3,4,5,6]Department of Mechanical Engineering, Annasaheb Dange College of Engineering & Technology, Maharashtra, India.

Abstract: The aim of the project is to create the plastic clip by using CATIA V5 R20. The part modeling, Core-cavity design, CNC manufacturing programming. The material selection for mold design is taken as Mild Steel. Cost of the total die assembly and cost comparison of different plastic components (HDPE, ABS, PP, and PC) are estimated. Here the process is using in injection molding and manufacturing a variety of parts from simple to complex components.

Despite several advantages of plastic injection molding, the process of manufacturing an injection mold tool is still a complex and highly skilled task that is very costly. Once the design is confirmed it usually takes several weeks or months to actually manufacture and market the product. This is mainly due to the complexity involved in creating the mold tooling. The purpose of this project is to develop a cost effective injection molding method for plastic clip.

This project deals with Design and Assembly of Injection Molding Die of Plastic clip of school Bag. The injection molding die is designed by applying proper design procedure.

Keywords: *Catia V5 R20 Modeling -Plastic Injection Molding-Manufacturing Technique -Catia Models Of Cope, Drag, Spacer, Bottom Plate, Ejector Plate, Ejector Back Plate, Ejector Pin-Catia Assembly Model,Design Of Gate, Runner-Defects.*

I. INTRODUCTION

The use of plastic is increased now days in many industries like automobile, packaging, medical, etc. The reason behind this is that the plastic made things are quiet easier to manufacture, handle and reliable to use. This project deals with Design and Manufacturing of Plastic Injection Mold of Plastic Clip. This is a component is used for school bags. It holds the strip of bag. The injection mold die is designed by applying proper design procedure.[1]

Despite several advantages of plastic injection molding, the process of manufacturing aninjection mold tool is still a complex and highly skilled task that is very costly. Once the design is confirmed it usually takes several weeks or months to actually manufacture andmarket the product. This is mainly due to the complexity involved in creating the moldtooling. The purpose of this project is to develop a cost effective injection molding method for plastic clip.[3]

Injection moldingis a manufacturing technique for making parts from both thermoplastic and thermosetting plastic materials in production. Molten plastic is injected at high pressure into a mold, which is the inverse of the product's shape. After a product is designed, usually by an industrial designer or an engineer, molds are made by a Mold maker (or toolmaker) from metal, usually either steel or aluminium, and precision machined to form the features of the desired part.[2][4]

Injection molding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of cars. Injection molding is the most common method of production, with some commonly made items including bottle caps and outdoor furniture. Injection molding typically is capable of tolerances equivalent to an IT Grade of about 9–14.

The most commonly used thermoplastic materials are polystyrene (low cost, lacking the strength and longevity of other materials), ABS or acrylonitrile butadiene styrene (a terpolymer or mixture of compounds used for everything from Lego parts to electronics housings), polyamide (chemically resistant, heat resistant, tough and flexible – used forcombs), polypropylene (tough and flexible – used for containers), polyethylene, and polyvinyl chloride or PVC (more common in extrusions as used for pipes, window frames, or as the insulation on wiring where it is rendered flexible by the inclusion of a high proportion of plasticiser).[9][14]

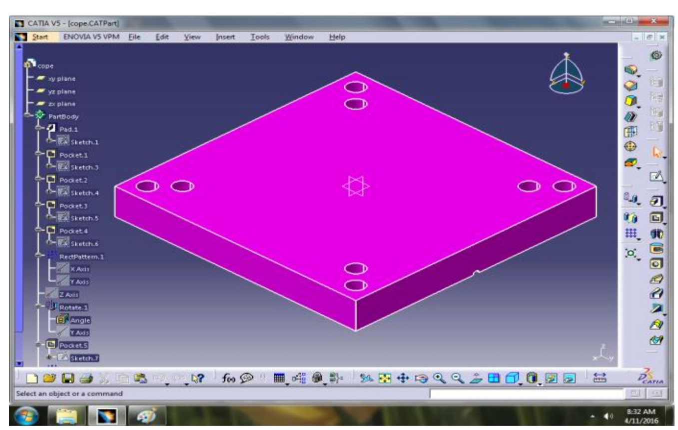
A. Objective of the Experiment

- To prepare a product design for Plastic clip by using design procedure.
- To manufacture Low cost Injection Molding Die.
- To test the plastic injection mold by using Hydraulic injection molding machine.
- To utilize for home business.

II. DESIGN

A. Cope

The top plate material is M.S. The size of top plate is 150 X 150 X 20. The Top Plate is used to locate the runner and convey the molten plastic material. The cope is as shown in picture 2.1.

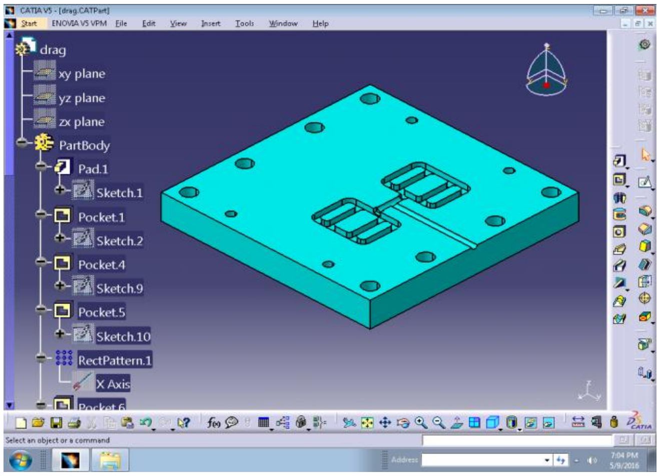


Picture 2.1: CATIA Model of Cope

B. Drag

The basic mold in this case consists of two plates. Into which one plate is the cavity which shapes the outside form of the mounting and is therefore known as cavity plate.

The Die Plate material is M.S. The size of top plate is 150 X 150 X 20. The Die Plate is used to locate the runner and convey the molten plastic material. The actual product shape is machined on die plate as shown in picture 2.2. Die plate has ejection holes as well as clamping holes.

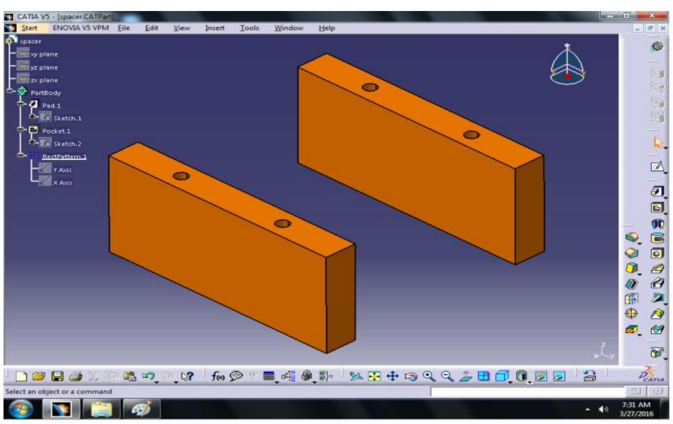


Picture 2.2: CATIA Model of Drag

C. Spacer

Spacer is for maintaining the gap between bottom plate and core back plate and also it helps in maintaining the required mold height.

The Spacer Plate material is M.S. The size of spacer plate is 150 X 20X 70. The spacer Plate is used to clamp the drag plate & bottom plate. The ejection stroke is depend on the height of spacer plate. Spacer plate is as shown in picture 2.3.

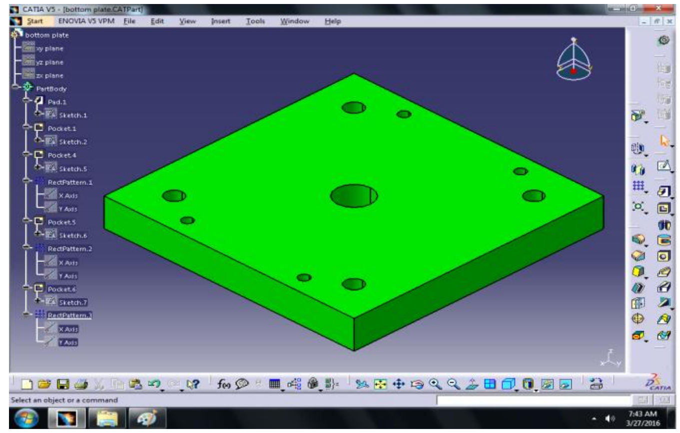


Picture 2.3. CATIA Model of Spacer

D. Bottom Plate

The bottom plate houses the clamping screw the main purpose of bottom plate is to clamp the movable half of the mold together i.e. core plate, core back plate, spacers and ejector plate.

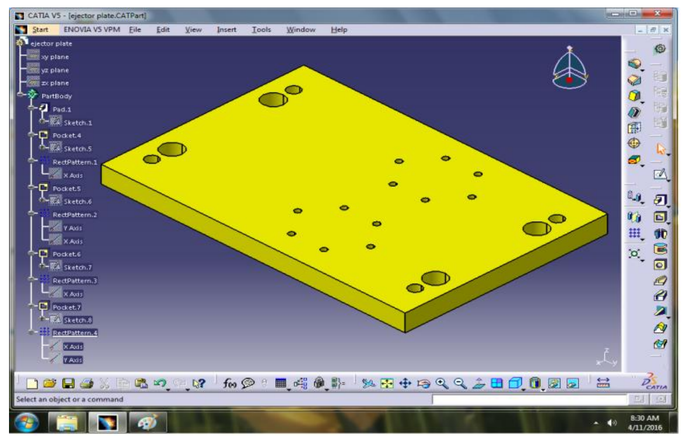
The Bottom Plate material is M.S. The size of Bottom plate is 150 X 150 X 20. The whole assembly is mounted on bottom plate. The ejector plate is rest on theBottom Plate. Bottom plate is as shown in picture 2.4.



Picture 2.4: CATIA Model of Bottom Plate

E. Ejector Top Plate

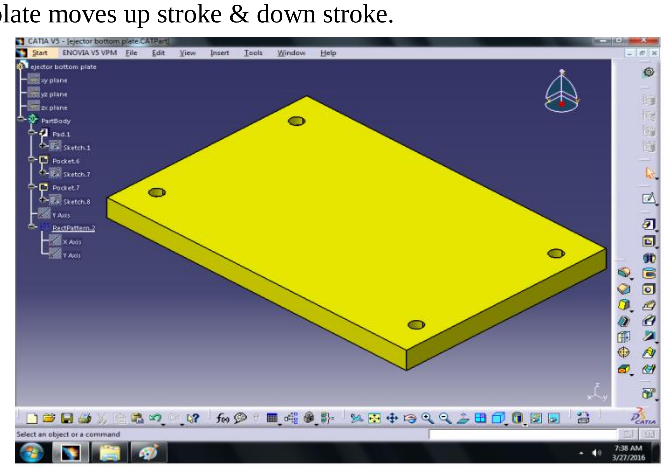
The Ejector Plate material is M.S. The size of Ejector plate is 125 X 40 X 15. The ejector pins clamps on the ejector plate. Ejector plate is as shown in picture 2.5. The ejector plate moves up stroke & down stroke.



Picture 2.5: CATIA Model of Ejector Plate

F. Ejector Back Plate

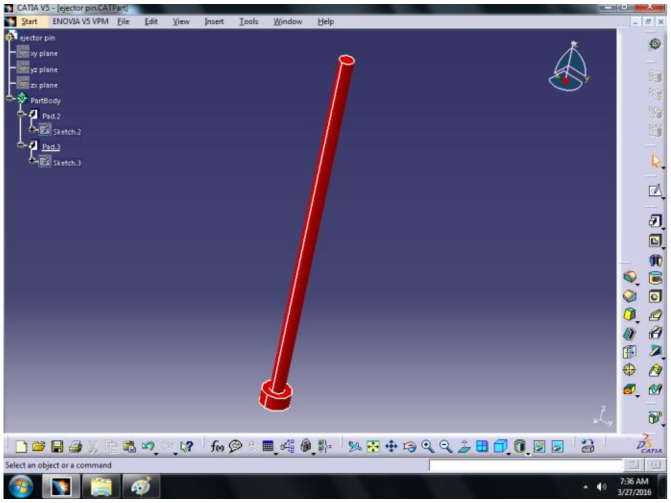
The Ejector Plate material is M.S. The size of Ejector plate is 125 X 40 X 15. The ejector pins clamps on the ejector plate. Ejector back plate as shown in picture 2.6. The ejector plate moves up stroke & down stroke.



Picture 2.6: CATIA Model of Ejector Back Plate

G. Ejector Pin

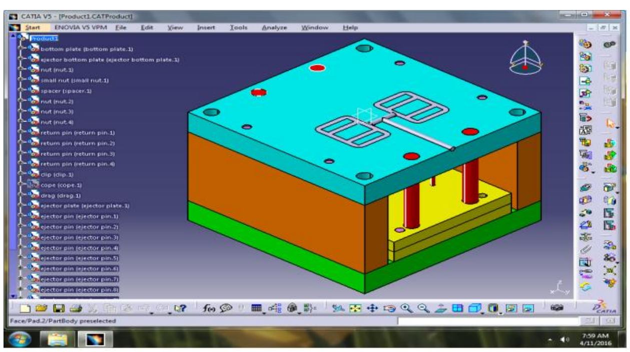
The Ejector Pin material is M.S. The size of Ejector plate is Dia. 3 X 72. The ejector pins clamps on the ejector plate. Ejector pin is as shown in picture 2.7. The ejector pin ejects the sample up side.



Picture 3.8: CATIA Model of Ejector Pin

H. Assembly

After assembly the Injection Molding Die of plastic clipis as shown in picture 2.8.[13]



Picture 2.8: Assembly

I. Design of Gate

The size of the gate can be considered in terms of gate cross section area and gate length. The optimum size for gate will depend on,

i. Flow characteristics of the material to be molded.
ii. The wall section of the molding.
iii. The volume of material to be injected into impression.
iv. The temperature of melt.
v. Temperature of mold.[14]

J. Design of Runner

When deciding the size of the runner following must be considered:

i. The wall section and volume of molding.
ii. Distance of the impression from the main runner.
iii. Runner cooling type used.
iv. The range of cutters available.
v. Plastic material that being used.
vi. The cross sectional area of the runner must be sufficient to permit the melt to pass through and fill the impression before the runner freeze and for packing pressure to be applied for shrinkage compensation if required.

Calculation of runner size:

$$D = \frac{W^{0.5} * L^{0.25}}{3.7}$$

Where,
D = runner diameter.
W = weight of molding.
L = length of molding.[7][8]

III. RESULT & DISCUSSION

A. Material Selection

Molds have been expensive to manufacturer. They were usually only used in mass production where thousands of parts were being produced. Various types of materials for manufacturing a mold are as follows:

1. Hardened Steel
2. Pre-hardened Steel
3. Aluminum
4. Beryllium-Copper Alloy
5. Mild Steel

Conclusion:

The first 4 materials are expensive as well as used for mass production. And machining cost of this materials is also high. For home business &Batch production we can use mild steel because properties of all the materials are nearly same.

B. Defects Occurred & Its Remedies

1. Molding Flash:

Molding flash occurs when a thin layer of material is forced out of the mold cavity at the parting line or ejector pins location. This excess material remains attached to the molded article, and normally has to be manually removed. The defect occurred is shown in picture 3.1.

Picture 3.1: Mold Flash Defect

Causes:

1. Worn or poorly fitting cavity/mold plates including, mold plate deformations and obstructions (grease, dirt, debris)

2. Insufficient clamp force the machine clamp force must be greater than the pressure in the cavity (that is, clamp opening force), to sufficiently hold the mold plates shut.
3. Over-packing- Over-packed sections causes increased localized pressure.
4. Non-optimal molding conditions-Including material viscosity, injection rate, and runner system. For example, high melt temperature, which makes a less viscous melt.
5. Improper venting-An improperly designed venting system, a very poor venting system, or a venting system that is too deep.

Remedies

1. Ensure correctly fitting mold plates- Set up the mold to seal properly. Clean the machine from any obstructions. Add pillar support or thicken the mold plates if there is any deformation of the mold plate during the molding process.
2. Avoid over-packing
3. Select machine with higher clamp force
4. Vent appropriately-Use the material supplier recommended venting size.
5. Optimize processing conditions-Reduce pressures and shot size to the minimum required.

2. Molding short shot

A short shot is the incomplete filling of a mold cavity which results in the production of an incomplete part. If a part short shots, the plastic does not fill the cavity. The flow freezes off before all of the flow paths have filled.To ensure the finished part is of good quality, the part must also be adequately packed with plastic.

Causes

1. Flow restrictions due to channels freezing or inadequate runner design.
2. Hesitation and long or complex flow paths.
3. Back pressure due to unvented air traps can cause a short shot
4. Low melt and/or mold temperature.

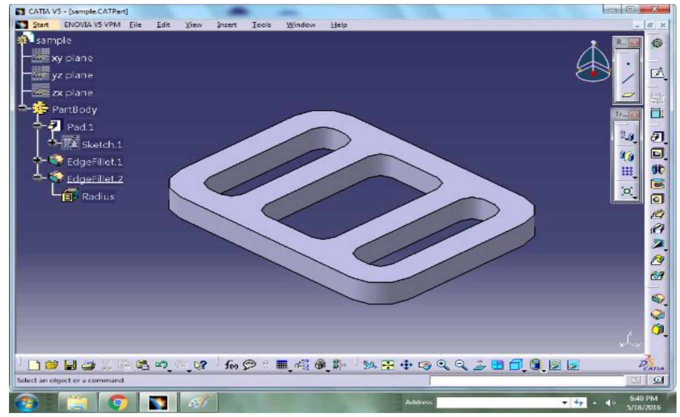
Picture 3.2: Mold Short Shot Defect

Remedies

1. Eliminate air traps.
2. Increase mold and melt temperature.
3. Increase ram speed.[14]

IV. FINISHED PRODUCT

The finished product is used in the school bags to hold the belts. The material of finished product is ABS (Acrylonitrile butadiene styrene). The finished product is look like as shown in picture 4.1 & 4.2.



Picture 4.1: Finished Sample (CATIA)

Picture 4.2: Actual Finished Sample

CONCLUSION

Injection molding has been a challenging process for many manufacturers and researchers to produce products meeting requirements at lowest cost. The presented work deals with the design and development of injection mold tool of a plastic clip for school bag. CAD/CAE technology facilitates the use of numerically controlled machining technology in manufacturing of mold. In, turns this reduces number and complexity of manual setup operations.

As can be seen from the above, the engineering and creation of injection molds is a time consuming process. The work is demanding in terms of knowledge, skills and exacting attention to details.

The designing was carried out with CATIA software. CATIA has been a real breakthrough in the industry by becoming the primary source of communicating design. Due to the use of the software the drawings need not be filed and stored in folder unlike in manual drafting and it is easily saved on the computer. 3D drawings are the best way to virtually represent a structure. Though one can manually create a 3D model it wouldn't look as realistic as the 3D model generated by CATIA. While creating any drawing on paper, there is bound to be some amount of revision or modifications. In manual drafting we need to erase and redraw to make any modification in the drawing but due to CATIA this rework and time required for it is get reduced. Location of gates is quite difficult in family molds, using mold flow analysis best gate location is selected. Also mold flow analysis is carried out to ascertain possible variations in plastic flow and process variables.

The Injection Molding Die of small plastic clip of school bag is most suitable for small size industries.The productivity is increased, manual labour required is less, the operation is simplified.

References

[1] Taylan Altan, Blaine Lilly, “Manufacturing of dies and molds”, The Ohio State University, Columbus, USA, 2010

[2] P. Krajnik, J. Kopac, “Modern machining of die and mold tools”, ELSEVIER, University of Ljubljana, Slovenia, 2004.

[3] Murlidhar Lakkanna, G.C. Mohankumar, “Design of moldsprue for injection molding thermoplstics”, ELSEVIER, NIT, Karnataka, 4th June 2015.

[4] MartonHuszar, “Sustainable injection molding: The impact of material selection and gate location on part warpage and injection pressure”, ELSEVIER, Warpage, College of engineering, Swansea University, UK, 2015.

[5] C.L.Li, “Automatic layout design of plastic injection mold cooling system”, The City University of Hong Kong, 2001.

[6] M.W.Fu, “The application of surface demoldability and moldability to side-core design in die and mold”, ELSEVIER, Core, Hong Kong, 2008.

[7] Hsien-Chang Kuo, Ming-Chang Jeng, “The influence of injection molding on tribological characteristics of ultra-high molecular weight polyethylene under dry sliding wear 268 (2010)”, 803-810.

[8] M.C.Song, “Research on effects of injection process parameters on the molding process for ultra-thin wall plastic parts journal of material processing technology”, 2007, 668-671.

[9] T. Boronat, “Influence of temperature and shear rate on the rheology and processability of reprocessed ABS in injection molding process”, ELSEVIER, ABS, 2008.

[10] H.S.Wang, “Cost estimation of plastic injection molding parts through integration of PSO & BP neural network”, ELSEVIER, 2004.

[11] All about plastic moulding. 2009. The History of Plastic Moulding. [ONLINE] Available at: http://www.plasticmoulding.ca/history.htm#Plastic

[12] H. Kurtaran, T. ERZURUMLU, “Efficient warpage optimization of thin shelled plastic part using response surface methodology and generic algorithm”, Adv. Manufacturing technology 27 (2006), 468-472.

[13] Manufacturer of Injection Molding Machine, Hikon, India, www.hikonindia.com/product-details.

[14] Design and manufacturing of plastic injection mold, 1-27

[15] The Rodon Group, Manufacturing Company based in U.S.A., info.rodongroup.com/blog/types.

[16] C. P. Natesha, K. Rameshbabu, “Design development and analysis of injection mold tool for divider housing median marker thermoplastic component”, 13th July 2012.