# Design And Development Of Mould For Wire Manager: A Review

P. B. Pimpalkar[1], V. S. Khangar[2],

1. M.Tech (CAD/CAM) Student Department of Mechanical Engineering, G.H. Raisoni College of Engineering, Nagpur - 440016 (India)

2. Assistant Professor, Department of Mechanical Engineering, G.H. Raisoni College of Engineering, Nagpur - 440016 (India)

***Abstract*: It is tedious to pass number of wires of different sizes through the slot provided in existing wire manager so, it is required to design the mould for wire manager and improvement in existing design is a must. An effort is made in this paper to review various existing techniques of designing the mould for wire manager. This will provides new direction to design and analysis of mould for multi slot on wire manager.**

## I. INTRODUCTION

Wire Manager is a wonderful tool for Wire Management as it manages so many wires without disturbing the beauty of the place. Wire Manager is a circular or ring type, inserted into the surface through a hole so that wires can be passed through it. It is generally used on working table, computer table, networking site and outdoor facilities. Wire manager consist of four parts top cap, wire manager body, rotating cap and Peg. Wire Manager plays an important role when wires of different type have to pass through it. Wire manager provide protection to the wires passing through it by covering all the sharp edges of the hole made on the surface. Existing wire manager is available with single slot on it so it is not helpful for multiple wire management. For making the mould of multi-slot on the wire manager various machines like EDM, VMC, CNC lathe machine, grinding machine are used. Pro/Engineer is a powerful CAD/CAM designing Software, which has been used extensively in various Production fields such as machines, automobiles, toys, and mold designing.PRO-E software is generally used for designing the injection mold. Tool path generation is carried out by POWER MILL software of dell

Cam. This paper review provides the construction and elaboration of mold for multi slot on wire manager.

## II. LITERATURE REVIEW

Based on Pro / Engineer software, cavity and core, the main part of the whole injection mold, can be designed, the entire designing process includes nine steps, which could be described as follows.
(1) Create a new file. Choose 'manufacturing' - 'mold cavity' in Pro / Moldesign module.
(2) Assembly the reference model of plastic parts, which is the basis for the establishment of the injection model. Direction and position of the reference model of plastic parts are connected to the mold-opening direction and the impression layout.
(3) Set shrinkage. According to the character of plastic materials to determine the appropriate shrinkage, the value is set to the reference model so as to modify the model. (4)Create workpiece. The workpiece is for the creation of mould cavity and core.
(5) Create the parting surface. The workpiece will be divided into separate components along the parting surface, which is very important to whether the cavity and core is developed successfully and is the key step in the design of injection mold.
(6)Divide the workpiece into separate volume blocks (mold components) by using previous parting surface. Segmentation methods may be different for different product structures.
(7)Extract mold parts-cavity and core. That means mold volume blocks divided in last step is extracted into mold parts that are modified in Pro/Engineering as an independent part. For the

parts, we may make further modification such as tensile, chamfer and fillet.
(8) Design feeding system.
(9) Simulate the mold opening and analyze the spatial location and movement of the mold elements, and detect whether interferences exist [1].

In order to satisfy the requirement of flexible production, injection mold designed and manufactured effectively and timely. Now a day, the development of injection mold is transferring from 2D mode to 3D. With the help of these 3D technologies, the design progress becomes more transparent. The whole design process including the mold cavity and core, feeding system, ejection system as well as cooling system and successfully conduct the simulation [1].

## 1.1] Cavity layout determination

Cavity & core are the main part of whole injection mould & it is design with the help of pro-e software. For the convenience of mold repair and replacement as well as cost savings, designer design individual mold parts for each impression. Fig.1shows the structure for mold element.

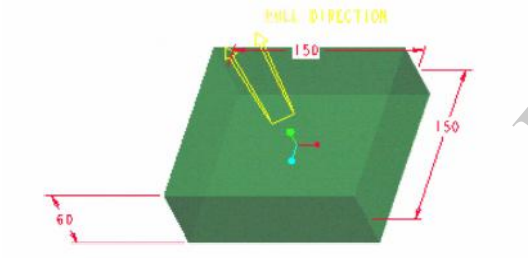



Fig. 1: structure for mold element [1]

## 1.2] Parting Surface determination

To determine the parting face, designers need to consider various factors such as the structural character and the appearance requirements of plastic parts as well as ejector methods. Fig. 2 shows the parting face for the whole mold.

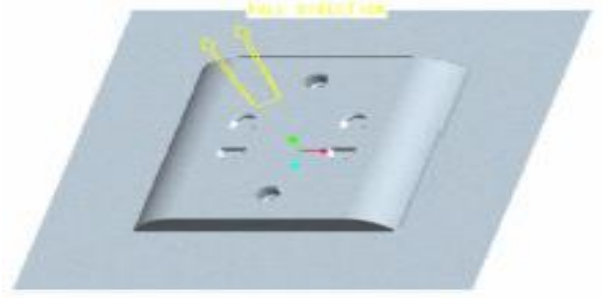


Fig. 2: parting face for the whole mold [1]

## 1.3] Molding Part Design

1] Design the structure of cavity

There are two types of cavity includes integer cavity and inserts cavity. When the cavity is machined from a large plate it is termed an integer cavity plate. Impressions and for multi- impressions molds, it is not satisfactory to attempt to machine the cavity from single blocks of steel as with integer molds. Instead of this the insert-bolster assembly method is used. It depends on product structure, production volume, mold costs and other factors. For the medium-size outlet, the requirement for the appearance is very high. So we choose the integral cavity for it has better stability and avoid leaving traces of mosaic. Fig.3 show the cavity structure.

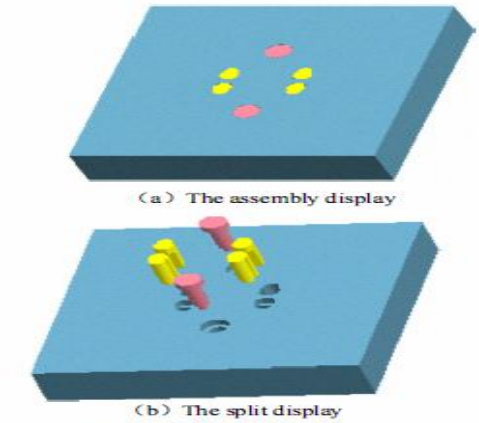
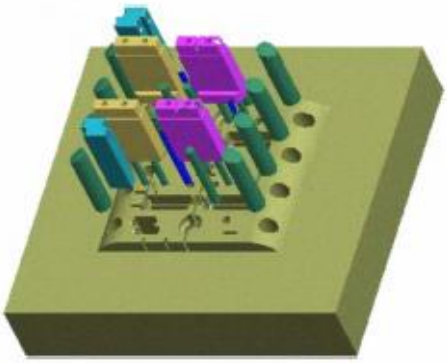



Fig. 3: The cavity structure [1]

2] Design the structure of cores

Cores are used to form the inner surface of parts or holes. Fig.4 shows the core structure.



(b) Split display

Fig. 4: The core structure [1]

## 1.4] Feed System Design

Design of feed system comprises the sprue, the runner, the gate and the cold slug.

1] Design the sprue

A sprue is a channel through which to transfer molten plastic injected from the nozzle of the injector into the mold impressions.

2] Design the runner
The runner, related to the cavity layout, is a channel that guides molten plastic to flow into the cavity of a mold.

3] Design the gate
The gate is a channel connected to the runner with the impression. The size of the gate can be considered in terms of the gate cross-sectional area and the gate length.

4] Design the cold slug well
The purpose of the cold slug well generally opposite the sprue, is used to receive the plastic material that has chilled at the front of the nozzle during the cooling and ejection phase. Fig.5 shows the feed system.

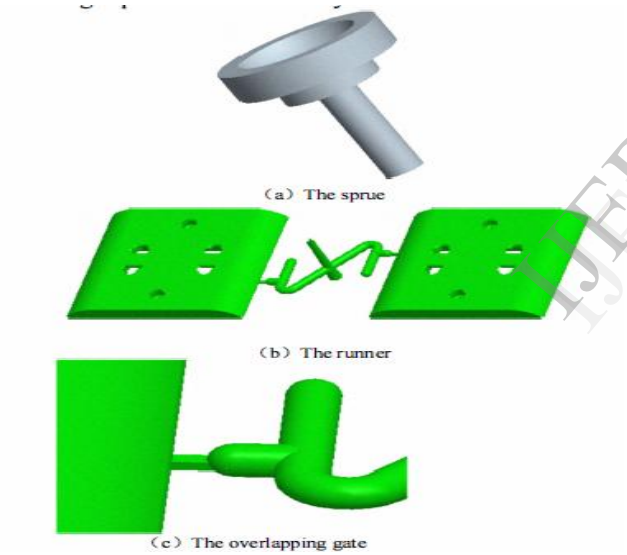
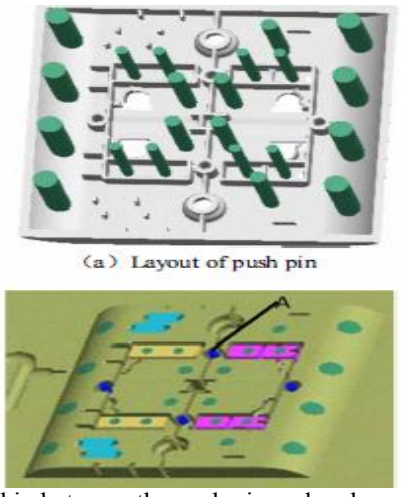



Fig. 5: The feed system [1]
[A] The sprue
[B] The runner
[C] The overlapping gate

## 1.5] Design the Ejector

There are several ejection techniques (such as pin ejection, sleeve ejection, stripper plate ejection and air ejection) that can be chosen while design the ejectors. Which technique to choose depends on the shape of the mold Parts. Here pin ejection type is choose. Fig.6 shows the push pin.




[b] Relationship between the push pin end and core
Fig. 6: The push pin [1]

## 1.6] Design Cooling System

The operating temperature for a particular mold will depend on a number of factors which include the following: type and grade of material to be molded; length of flow within the impression; wall section of the molding; length of the feed system.

1.7] Simulate Mold- Opening and create injection pieces
After complete initial design, we assign mold-opening simulation steps. Simulation of mold opening process can help to observe each mold element relative spatial positions and movement trajectories. Fig.7 shows the 3D design result.

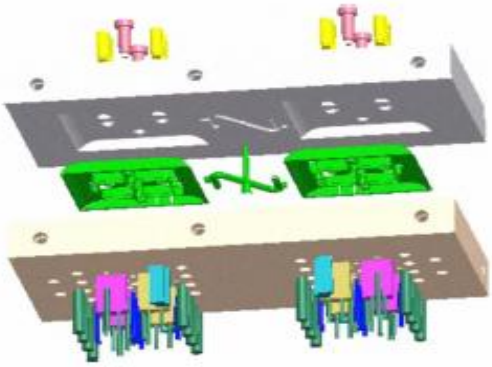


Fig. 7: 3D design result [1]

3D design results are more intuitive, easy to find design defects, and helpful to improve design accuracy and reduce the cost of mold and product development. This is the current development trend of injection mold designing. For the small and medium mold, failures of mold are often caused by lack of strength. Therefore, during the design, strength should be principally ensured [1].

In order to study lead MF elimination, new mould die design is established. The lead MF is excess epoxy mould Compound along the lead side it include resins, filler, hardener, flame resistance. It comes from the gap between the lead & the lead slot. When the mould compound fills into this gap, mold flash is formed. But this mould flash is undesirable, because it may cause some functional failure when it is placed on printed circuit board (PCB). The lead MF has two dimensions – length & thickness. The force required to remove the MF from the lead are product's package's compound strength, high pressure water force, and compound adhesion stress from lead frame [2]

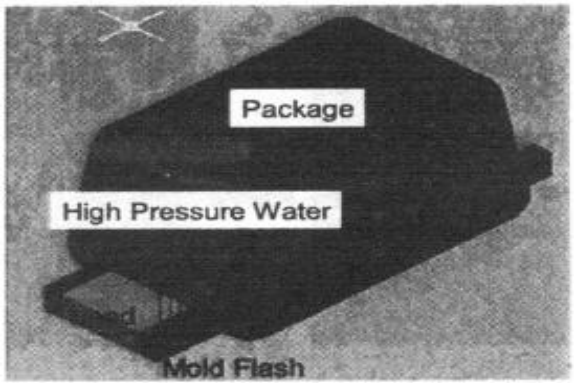



Fig. 8: Package structure after molding.

There are many methods by which flash length on packages can be removed, such as Mechanical Deflash, Laser Cutting Deflash and Ultra High Pressure Water Jet Deflash. But this method is ineffective in removing mold flash on micro miniature packages. Hence new mold die design is establish a predetermined break point to enhance elimination of excess lead mold flash and it is a combination of triangular and wing type concepts that will eliminate lead MF Defects
[2].

The parting surface is the surface that separates the mould insert into the core and cavity halves. A common strategy in parting surface generation is the method of extrusion. CAD/CAM systems are currently used in the injection mould design and manufacturing process. If all segments are visible in any one of the directions perpendicular to the mould inset boundary, the entire parting line is visible and the method of extrusion can be used. Plastic part and the main parting line, a common approach to generate the parting surface is illustrated in Fig.9

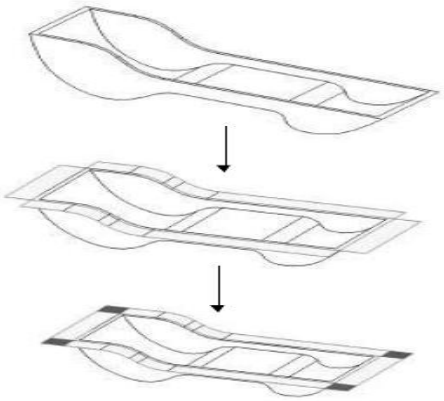


Fig.9: Generating parting surface by method of extrusion [3]

But this method fails under certain conditions, which results in intersection between the parting surface patch and the part surface, inter-locking between the mould halves & it affects on mould opening. Fig.10 shows the method of extrusion causes inter-lock between mold halves.

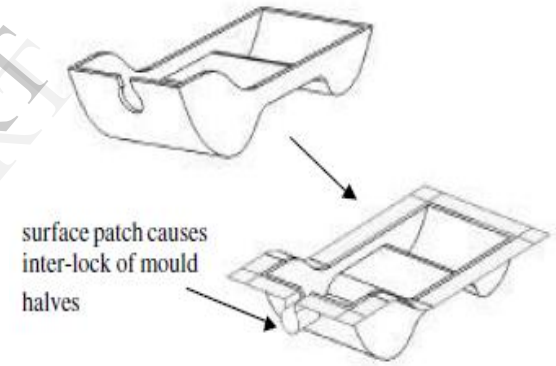



Fig. 10: The method of extrusion causes inter-lock between mold halves. [3]

To solve these problems, a new method based on the subdivision method of surface generation is proposed. A variation of the Cat mull-Clark subdivision scheme is employed which ensures that the resulting surface interpolates the main parting line of the part. The first step in the generation of a subdivision surface is to construct the initial control mesh. The initial mesh can be considered as the skeleton of the final surface. To determine this 3D skeleton, a 2D skeleton is first determined, which can be obtained by computing the medial axis of an approximating polygon. The 3D mesh is then obtained by "projecting" the 2D skeleton to 3D. Using this method, the subdivision surface can be made to interpolate a set of curves that defines the boundary of the surface. This is an important property as the surface patch of the parting surface must attach to the parting line segment. For the

invisible segments, a new method based on the Catmull-Clark subdivision method is used [3].

RAPID TOOLING is used to produce temporary molds as well as permanent molds. The objective of this research is to develop the direct metallic RT system and identify process parameters of laser cladding. The two-step process involving an intermediate step of mold preparation in conventional RP techniques are

1] A pattern is made with RP techniques, from which a mold is created.

2] Laser-based RT systems have been developed to create functional, metallic prototypes with near net-shape geometries.

The drawbacks for these systems are their low productivity and inability to consistently regulate component quality in terms of mechanical properties and geometry. RT is capable of generating core and cavity inserts as a product of an RP technology. RT is the result of an additive process driven by 3-D CAD process that requires little or no

Machining or electric discharge machining (EDM).

The advantages of the direct metallic RT system include: shortening of the tooling lead time; reduced cost; functional test of parts in early design becomes possible; and an ability to directly transfer CAD data [4].

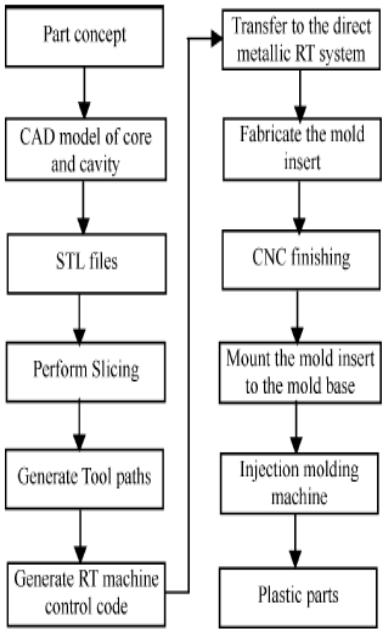



Fig.11: Flowchart of the development of injection mold with the direct metallic RT system [4].

Plastic products are manufactured by injecting the molten plastic into mould and then rapidly cooled down to get the desired parts. For a complex injection mould, there are more

than 100 geometry regions that should be manufactured by EDM. UG/NX Electrode Design Software is used to speed up the EDM electrode design process that supports automatic recognition of the EDM. Processing method is used to determining the extracted features. The extracted concave manufacture features usually include holes, rectangular sockets, and irregular sockets. Fig.13 shows D is the minimum distance between the closest points in the profile and T is the depth of the socket.

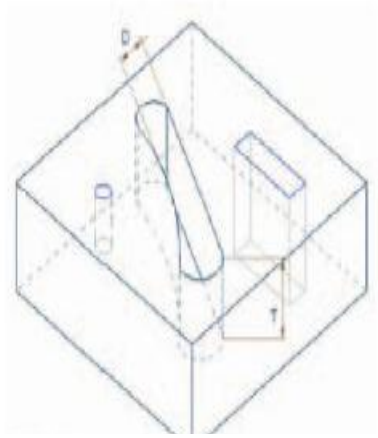


Fig. 12: Typical concave manufacturing features in mold. [5]

The material hardness also has important influence on processing method determination. If the material hardness is over 64 HRC, the mould must be produced by EDM because CNC milling does not provide the economical conditions for such case yet.

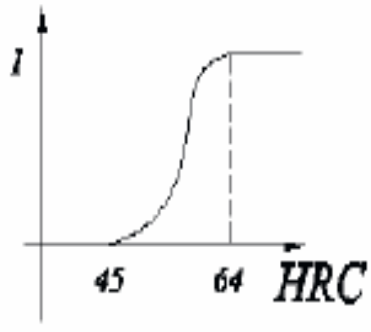



Fig. 13: Membership function of HRC [5]

A simplified procedure of processing method determination is described as following:

1. Get geometry parameters, such as *D* and *T*, from the extracted feature.
2. Get the material information and tool accessibility information.
3. Calculate the membership grade of each feature to the sets *N_*, P_, *C, H_* and *M.*
4. Calculate the membership grade to fuzzy set *F* [5]

**TABLE I**
Solve example of processing method determination

| SN | D(m) | $U_N$ (X) | T(mm) | $U_P$ (X) | Tool Accessibility | $U_C$(X) | hardness |
|---|---|---|---|---|---|---|---|
| 1 | 120 | 0.09 | 30 | 0.22 | 1 | **0** | 60 |
| 2 | 4 | 1 | 2 | 0 | 1 | 0 | 40 |
| 3 | 12 | 0.99 | 30 | 0.22 | 1 | 0 | 55 |
| 4 | 12 | 0.99 | 40 | 0.78 | 1 | 0 | 55 |
| 5 | 120 | 0.09 | 30 | 0.22 | 1 | 0 | 65 |
| 6 | 120 | 0.09 | 30 | 0.22 | 0 | 1 | 40 |

CAD and CAM programmed is used for technical documentation-drawings and also for cutting technology planning in moulds machining. CAM (Computer Aided Manufacturing) programmed for mould machining is prepared for application on 3-axial milling CNC machine tool. Fig. 14 shows the product sustainability.


Fig.14: The product sustainability [6]

Parametric modeling is very useful and above all very important for sustainable production achieving. Because we can, with some minimal geometric corrections, use the already constructed and perfected model for the elaboration of new models. The tool geometry usually remains the same, in some defined intervals only the tool or product dimensions change. Of course it is very important to achieve the perfect tool construction and the perfect modeling parameters. The principal condition for parametric tool modeling is a repeated tool use. This condition is also principle for sustainable production and sustainable products achieving. Tool making continues with the planning of CAM and the final tool making on a CNC machine-tool. 3-axis machining performed on a CNC machine-tool can be indicated as a sustainable procedure [6].

Core-pulling mechanisms are the most important in the designing of the injection mould. The action of the core-pulling mechanisms, ejector mechanisms and resetting mechanisms is stable and reliable. Core-pulling mechanisms are designed reasonably in one mould and demolding of the plastic parts is realized. Fig. 15 show the angle pin core-pulling mechanism

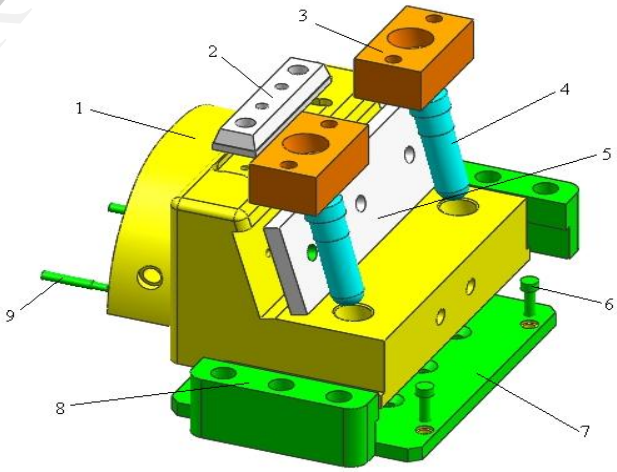



Fig.15: Angle pin core-pulling mechanism [7]

Description of parts is as bellow.

1. Slide
2. Fixed block
3. Locking block
4. Angle pin
5. Wear plate
6. Positioning screw
7. Wear plate
8. Slide wedge
9. Spring rod

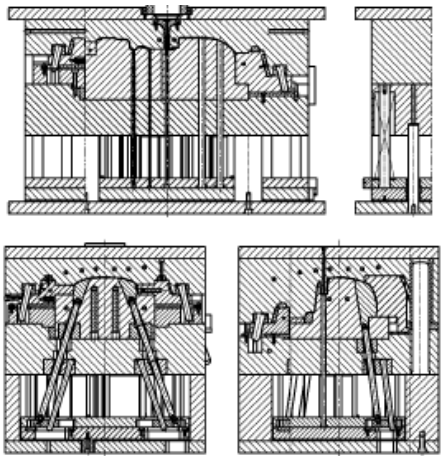


Fig.16: Mold structure

Mould structure shows the core pulling mechanism [7].

Cad software is most widely used in mechanical industries. For mould & die manufacturers CAM, CAE, CAPP, Collaborative design methods are used. Now a days CSCD is used in mould & die industry to shorten delivery time & improve product quality. XML driven design is proposed as a new method that it aims to provide theories & techniques, methodologies to support design, automation & management of mould & die design process. XML is used to support design reuse for mould & die design. It consist of four models

1) CAD platforms
2) Design event DB
3) XML schema
4) Parse & tools [8].

Fig. 17 show the proposed framework of XML driven design mechanism.[8]

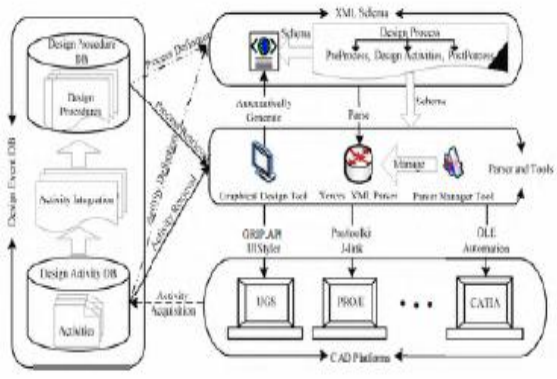



Fig. 17: The proposed framework of XML driven design mechanism. [8]

## III. CONCLUSION

In this literature review the researcher has carried out complete design process related to the mould and gives brief idea about injection molding method. Researcher used CAD/CAM, Pro-E software for making the mould. But until now research is not carried out on the designing of mould for wire manager and today, there is a need of mould for multislot on wire manager for proper wire management. So ,now research will be carry for making the mould for wire manager for that various types of machines like grinding machine, electric discharge machine ,vertical machining center , CNC milling machine and some advanced technologies , such as CAM ,CAE ,Collaborative design methods are use. With the help of this techniques mould will be prepared and it will help to improve design quality of mould and gives shorten delivery time. ABS materials are used for wire manager. After finishing the mould for wire manager it is clamped into the injection molding machine that generates the wire manager.



## REFERENCES

[1] Ruan Jianhua, Wang Yue, Zhang Jingyuan, Jerry Quah, “Molds die design to improve flash remains for micro miniature pacakage”IEMT, 2006.

[2] Cao Suhong, "3D Mold Design of Outlet Based on Prof Engineering” IEEE, 2012.

[3] CL Li, “Application of Catmull-Clark Subdivision Method in Plastic Injection Mould Parting Surface Design” IEEE, *2002*.

[4] Ren C. Luo, Jyh Hwa Tzou, “The Development of Direct Metallic Rapid Tooling System”IEEE, 2007.

[5] Chi Zhou, JianJun Liang, SaiJun Zhang, “A Fuzzy Approach to Determine Processing Method in Mould Manufacturing**”** ICINIS, 2009.

[6] Janez Kopac, Joze Balic, Franc Cus “Sustainable 3D Mould Design and Manufacturing” PICMET, 2008.

[7] Huang Guijian, Li Xuemei, Wu Xiaoyu, Li Jibin “Application of Core-pulling Mechanism in Injection Mould Design” IEEE, 2009.

[8] Shuangrong Yang, Mingyue Huang, Sha Hu, Jianjun Li, “Realizing XML Driven Design Activity for Mold & Die Design” IEEE ,2010.