# Injection Mold Development Using Unigraphics as CAD Software for Mass Scale Production of a Plastic Container

Parag G. Chavhan
*M.E. Student , Production Engineering*
*A.C.O.E. Sangamner, Maharashtra, India*

Prof. B.R.Borkar
*Professor*
*A.C.O.E. Sangamner, Maharashtra, India*

**Abstract – The aim of this paper is to present a study of work done in the project work on the injection molding process & injection molding die development. For this Unigraphics is used as Cad software. Injection moulding is a manufacturing process. It produces parts from thermoplastic and thermosetting plastic. Material is fed into a heated barrel, mixed & forced into the cavity of a mould. There it solidifies during cooling. It takes shape of cavity. Once a product is designed mold makers prepare mould from metals like steel, aluminium. Molds are manufactured either by standard machining or Electrical Discharge Machining. Now a days CNC machines are used for machining of these moulds. This paper presents study of injection moulding process & its mold development process for mass scale production of a small plastic container.**

**Keywords- Injection Molding, Injection Mold Design, Mold wizard, increasing no. of cavities of injection mold.**

## I. INTRODUCTION

Since 1940 due to World War II, injection moulding process got very high demand. It gives a very wide range of products like buttons, combs, automotives, toys, packagings, one piece chairs, mechanical parts like gears etc. This is suitable process for producing high volume of products. Other benefits are repeatable high tolerances, ability to use wide range of materials, less labour costs, lesser scrape, lesser finishing work. Disadvantages are costly equipments, higher running costs etc. For mold developments using Cad/Cam tools steps are to prepare 3D model of product component, then develop die cavities, then die blocks, then other parts of die assembly in 3D model form. Data of this 3D model helps to do Cam operations. To get final product component with good finish & close tolerances design of mould features carefully, is important. To improve the productivity(cost & time wise) there are many factors to focus, like reducing set up time, cycle time, waste, scrape, rejection, trials, manufacturing lead time, labor cost per piece etc. All these are much more related to injection mould structure & hence its design. One of the factors to improve productivity is to increase the no. of cavities of mould. It increases cost of manufacturing (& maintenance to some extent) of mould. But for mass scale production it offers more no. of pieces per unit time. This helps to improve labor & machine productivity to a great extent. In this project an injection mold is designed for producing a container. 2D part drawing is as shown in figure1. From part drawing 3D model of container is developed. Then 3D model of injection mould is developed step by step. For this Unigraphics is used as a Cad software tool. Mould wizard is the in-built facility to produce mould set in Unigraphics.

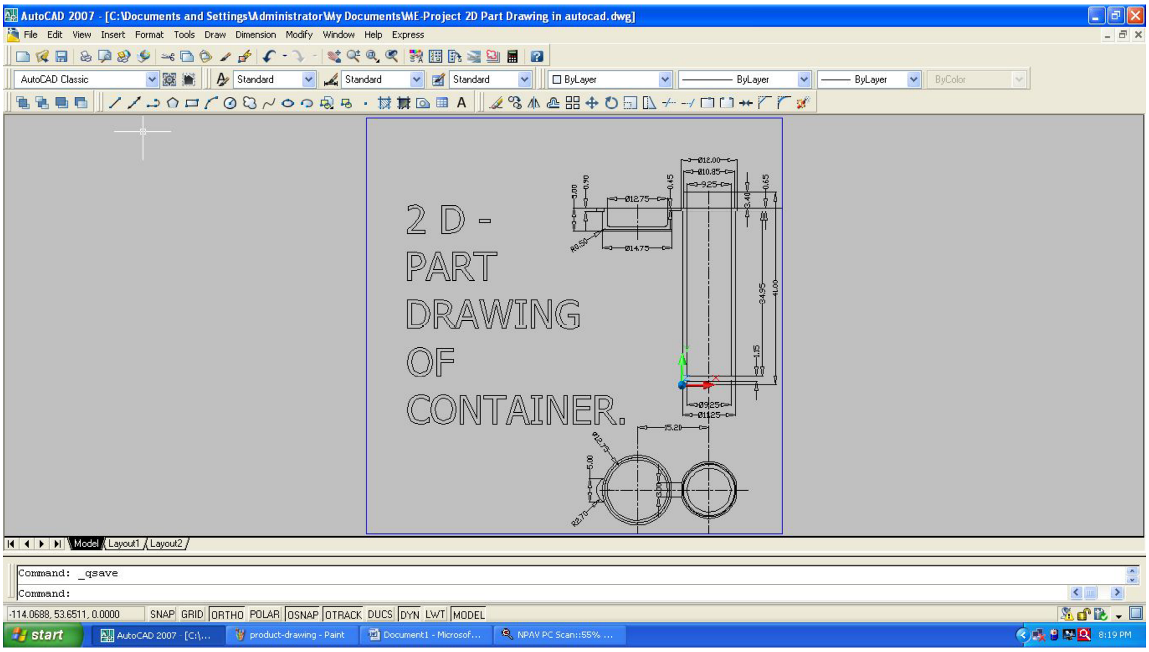



Figure1. Product Component: Container 2D-Part Drawing.

## II. LITERATURE SURVEY

“AN INTERACTIVE CAD/CAE SYSTEM FOR MOLD DESIGN” [1] by Ivan Matin, Miodrag Hadzistevic, Janko Hodolic, Djordje Vukelic. This work presents a knowledge-based parametric design system for mold design. It requires only a minimum set of injection molding parameters to be established before being able to complete the design of the main components of a mold. This CAD/CAE system contains of Pro/E, special developed modules to calculate, select, modify and design. Developed system can greatly improve the quality of design while reducing the development cost and time.

“A GENERIC FRAMEWORK FOR RAPID DEVELOPMENT OF INJECTION MOLDS AND PRESSURE DIE CASTING DIES” [2] by Nagahanumaiah, Bhallamudi Ravi, Prasad Mukherjee. This paper presents Rapid Product Development (RPD) a strategic solution to stay in global competition is characterized as the process chain that ensures development of flexible products in lesser lead-time at competitive cost. Even though die and mold development is recognized as a major step, tooling development procedure is not yet very well documented and is experience based, resulting in higher cost and longer lead-time. A research attempt has been made to develop a generic framework for rapid development of dies and molds, which comprises the methodologies for entire tooling development process starting from tooling requirements modeling, process modeling to rapid hard tooling development. This paper presents the methodology developed for each stage particularly for injection molding and pressure die casting. The methodology can be easily customized for any tool room, enabling reduction in lead-time and cost through minimization of tooling iterations and backflows, demonstrated by industrial examples. Implementation of this framework is proposed for future tool rooms.

“An Intelligent Cavity Layout Design System for Injection Moulds” [3] by Weigang Hu and Syed Masood. This paper presents the development of an Intelligent Cavity Layout Design System (ICLDS) for multiple cavity injection moulds. The system is intended to help mould designers in cavity layout design at concept design stage. The complexities and principles of cavity layout design as well as various dependencies in injection mould design are introduced. The knowledge in cavity layout design is summarized and classified. The paper also discusses such issues as knowledge representation and case-based reasoning used for the development of the system. The functionality of the system is illustrated with an example of cavity layout design problem.

“Knowledge – based method for gate and cold runner definition in injection mold design” [4] by Z. Rutkauskas, A. Bargelis. This paper deals with the application of intelligent system for injection mold design. The molding system of the developed knowledge-based module for injection molds in more detail is considered. For this aim the known theoretical methods of gates and runner’s definition as well as facts and rules were systematized considering experts’ knowledge & experience. The calculation accuracy of gates and runners were tested by the developed knowledge-based method comparing it with practical data. The developed KB method for gate and cold runner design is linked to the artificial intelligent area. The purpose of mentioned KB is to replace routine work of human experts by artificial intelligent methods. This would solve gate and cold runner’s definition problem of injection mold design. There are several opinions why it is in use e.g. for the purpose to decrease the amount of routine work, the beginning engineer in the field could be prepared in less time to use injection molds design methodology as expert can’t always be in his or her workplace, the working area of expert became complex. The advantage of KB is that the data, which expert imputed into data base (DB), can always be verified and supplemented reducing serious misunderstandings. Inference engine and knowledge base offer solutions to the user via user interface.

“Knowledge Engineering and Capitalization for Injection Mold Design” [5] by Suthep Butdee. This paper says that the process of Knowledge Engineering plays an important role in managing industrial knowledge. By its nature, industrial knowledge is combined and linked together as a chain. Industrial knowledge that is considered “good”, i.e., knowledge that is ensured to be complete and usable, must be validated by human experts. It is prior to both its initial and recurring use. This paper proposes the concept of knowledge capitalization associated by PCA method and validated by experts on shop floor.

“New Cooling Channel Design for Injection Molding” [6] by A.B.M. Saifullah, S.H. Masood and Igor Sbarski. This paper presents about injection molding which is one of the most versatile and important operation for mass production of plastic parts. In this process, cooling system design is very important as it largely determines the cycle time. A good cooling system design can reduce cycle time and achieve dimensional stability of the part. This paper describes a new square sectioned cooling channel system for injection molds. Both simulation and experimental verification have been done with these cooling channels system. Comparative analysis has been done for an industrial part, a plastic bowel, with conventional cooling channels using the Mold flow simulation software. Experimental verification has been done for a test plastic part with mini injection molding machine. Comparative results are presented based on temperature distribution on mould surface and cooling time or freezing time of the plastic part. The results provide a uniform temperature distribution with reduced freezing time and hence reduction in cycle time for the plastic part.

“3D RAPID REALIZATION OF INITIAL DESIGN FOR PLASTIC INJECTION MOULDS” [7] by Maria L.H. Low and K.S. Lee. This paper describes to provide an initial design of the mould assembly for customers prior to receiving the final product. CAD data is a preliminary work of any final plastic injection mould design. Traditionally and even up till now, this initial design is always created using 2D CAD packages. The information used for the initial design is based on the technical discussion checklist, in which most mould makers have their own standards. This technical discussion checklist is also being used as a quotation. This paper presents a methodology of rapid realization of the initial design in 3D solid based on the technical discussion checklist, which takes the role of the overall standard template. Information are extracted from databases and coupled with the basic information from customer, these information are input into the technical discussion checklist. Rules and heuristics are also being used in the initial mould design. A case study is provided to illustrate the use of the standard template and to show its real application of rapid realization of the initial design for plastic injection moulds.

“Set-Up Reduction in Injection Molding Process – A Case Study in Packaging Industry” [8] by B.Kayis, S. Kara. This paper states that the competitiveness of manufacturers can be significantly enhanced through implementation of Setup Reduction (SUR) initiatives. This paper presents a simplified SUR approach which was trialed and fully implemented in an injection molding facility. The identification of bottlenecks in production was carried out by using extensive data gathering on machine down-time and/or changeover time records. The detailed analysis of the operations carried out by employees was investigated by using the Single Minute Exchange of Dies (SMED) philosophy. Close collaboration with employees and formation of SUR teams facilitated the development of a system for the organization and storage of molds. The system developed ensures mold readiness conditions, reduces lead times and improves the storage, monitoring and accessibility of machine programs.

”Characterization of an Injection Molding Process for Improved Part Quality” [9] by A. Ghose, M. Montero, D. Odell. This paper describes that an injection molding process was characterized with respect to certain physical characteristics of an injection molded part. The produced part, the “Triconnector”, is a planar Lego-like piece, which was shot from ABS thermoplastic on a Morgan-Press G-100T Injection Molding Machine. Using statistical design of experiments, we identified the most significant parameters affecting part quality. The parameters used fell into three main categories: machine, process, and material. Part quality was defined by part fill, sink, and flash. Predictive models of part quality were developed and tested. It was found that ram speed, mold pre-heat, and nozzle temperature most affect part fill. Similarly, it was found that barrel temperature and cycle time most affect the sink of the sprue. A reliable predictive model for flash could not be developed, as there was too much variability in the results.

“Design and Simulation of Plastic Injection Molding Process” [10]. This paper presents the design of plastic injection mould for producing a plastic product. The plastic part was designed into two different types of product, but in the same usage function. One part is using clip function and another part is using tick function. In the computer-aided design (CAD), two plastic parts were drawn in 3 dimension (3D) views by using Pro-Engineer (Pro-E) parametric software. In the computer-aided manufacturing (CAM), Pro-Manufacturing from Pro-E parametric software was used to develop the machining program. For mould design, the product was designed into two changeable inserts to produce two different types of plastic product in one mould base. Before proceeding to injection machine and mould design, this part was analyzed and simulated by using Mold Flow or Part Advisor software. From the analysis and simulation we can define the most suitable injection location, material temperature and pressure for injection. The predicted weld lines and air trap were also found and analyzed.

## III. GEOMETRIC MODELLING

From 2D drawing 3D part model is developed. It is a solid model. It is shown in figure2. Part model represents geometrical details of part. From part model 2D views can be generated. We have used Unigraphics software for developing 3D model. Various commands used are in sketch mode as well as in 3D mode. 3D command used is extrude, revolve etc.

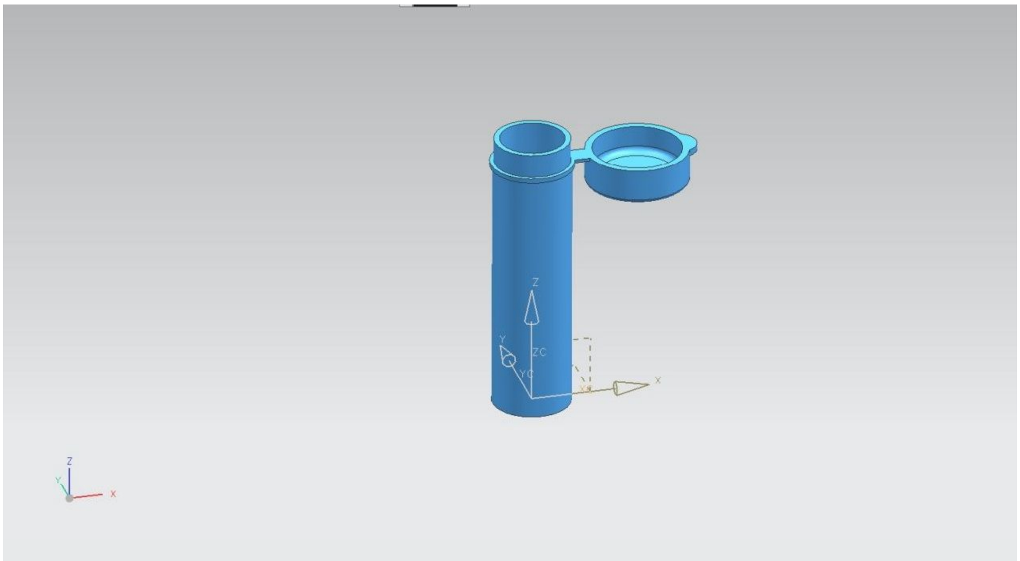
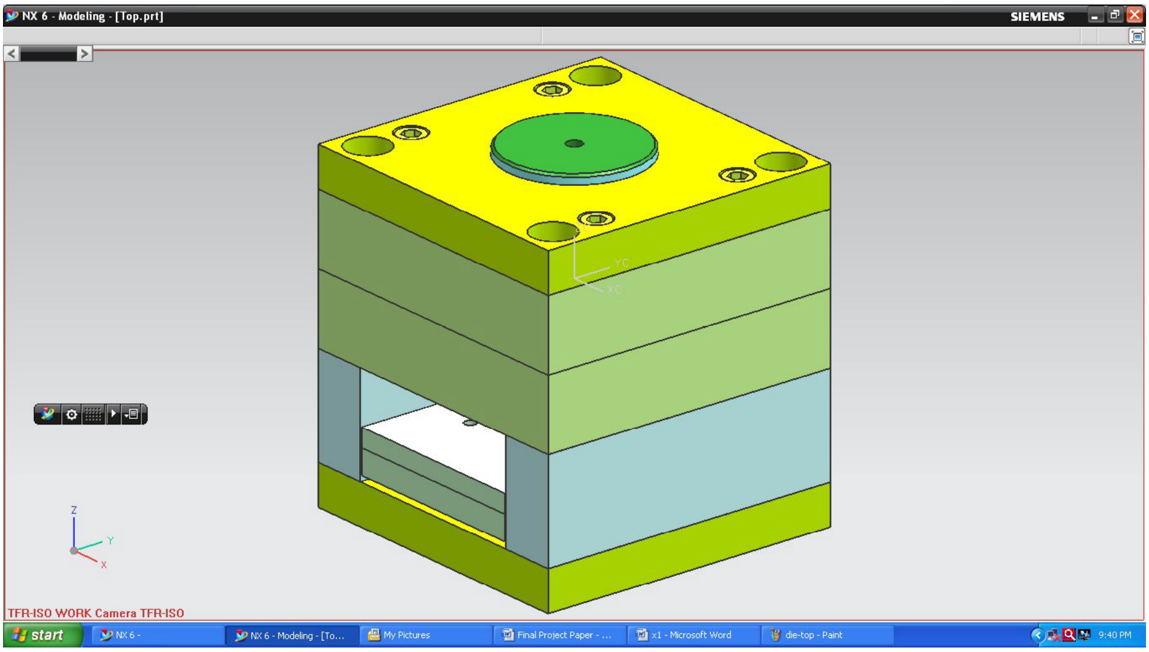


Figure 2. Product Component: Container 3D-Part Solid Model.

## IV. MODIFIED MOLD DESIGN

As a part of improvement in productivity it is expected to produce more no of parts in same time of interval. For this new design of mould is made. Previous system had two moulds. One mould has only one cavity of cylindrical part. Other had one cap cavity. New mould contains four no. of cavities. i.e. in one stroke four product components are produced each has cap attached with it. So average cycle time is reduced. There are certain factors to be considered while calculating no. of cavities to be kept on mould.

Figure 3. Complete mold set assembly.



For better understanding further in figure4 mold set is shown with some parts transparent so that inside details are seen.

## V. ANALYSIS

There are two achievements. One is that part is modified. It has cap with itself. So single mold is sufficient that is designed for manufacturing. This reduces mold production cost. Also average cycle time is reduced. Labor cost is reduced that of operating injection molding machine. Mould setting time on machine on average is also naturally reduced. No. of cavities are four. So this offers high production rate.

Comparison between Old & New mould(s) system:

Assumption is that one machine & one operator is under utilization in both systems.

A) Machine & Labor Productivity Improvement :

In old system:

1) Calculations for Cap mould :

Mould closing- opening time [2M] = 3 s.

Injection Time [T] = 2 s.

Cooling Time [C] = 5 s.

Ejection [E] = 2 s.

Weight of mould = 85 kg.

Cycle time formula is Cycle time TCT = 2M + T + C + E.

So Cycle time = 12 s.

Considering 10 hrs per day working = 3600 x 10 s per day = D say.

Max. no. of shots per day = D / TCT = 36000/12 = 3000 nos.

Therefore no. of caps produced per day = 3000 nos.

2) Calculations for Container mould:

Mould closing- opening time [2M] = 5 s.

Injection Time [T] = 5 s.

Cooling Time [C] = 6 s.

Ejection [E] = 4 s.

Weight of mould = 110 kg.

Cycle time formula is Cycle time TCT = 2M + T + C + E.

So Cycle time = 20 s.

Considering 10 hrs per day working = 3600 x 10 s per day = D say.

Max. No. of shots per day = D / TCT = 36000/20 = 1800 nos.

Therefore no. of containers produced per day = 1800 nos.

If only one machine is considered available & mould change time is neglected,

We solve equation Q x 12 + Q x 20 = 3600 x 10, where Q is the max target possible in this case (both molds sharing same machine & operator.

So Q = 1125 nos.

3) Calculations for new mould producing whole container with cap having 4 no. of cavities:

Mould closing- opening time [2M] = 5 s.

Injection Time [T] = 16 s.

Cooling Time [C] = 10 s.

Ejection [E] = 9 s.

Weight of mould = 350 kg.

Cycle time formula is Cycle time TCT = 2M + T + C + E.

So Cycle time = 40 s.

Considering 10 hrs per day working = 3600 x 10 s per day = D say.

Max. No. of shots per day = D / TCT = 36000/40 = 900 nos.

Therefore no. of containers produced per day = No. of shots per day x no. of cavities on mould.

Therefore no. of containers produced per day = 900 x 4 = 3600 nos.

B) Material Productivity Improvement: material consumed for gate & runners is different in both cases. But for thermoplastic material it is recirculated. So comparison study of material is not considered here for study of material productivity rise.

C) Cavity Layout: There were two options: circular & rectangular arrangement. Material flow is naturally balanced in all directions in circular arrangement. Steel we selected rectangular for achieving ease of manufacturing mould set parts. In rectangular options were like 2, 4, 6 etc. So 4 is chosen which can be arranged in square fashion for material flow balance equal around sprue axis. Maximum limit of no. of cavities depends on machine specifications. They are shot capacity, plasticizing capacity & clamping capacity. Each of these 3 factors give maximum's limit individually. Then out of these 3 values minimum is considered for design purpose.

Results: In previous case we could have production of 1125 nos. Using same resources of machine and labor hours, by modified mold we could have production of 3600 nos. Labor cost of operating machines is on hourly basis, remaining same in both cases. So rise in production per day = 3600 – 1125 = 2475 nos. So machine utilization is increased. This is helping improvement in productivity.

## VI. MOULD DEVELOPMENT USING MOULD WIZARD

There are steps to be followed to use Mould Wizard as given below:

1. Open product model file. Figure 2.
2. Goto start, all applications, mould wizard. Figure 5.
3. Select initialize project, specify material, type ok.
4. Then select Mold CSYS, Product Body Center, Lock z position, apply cancel.
5. Select Work piece, Product work piece, Work piece method –Cavity-Core, Cancel.

Figure 4: Mold assembly set with some parts semi-transparent state.

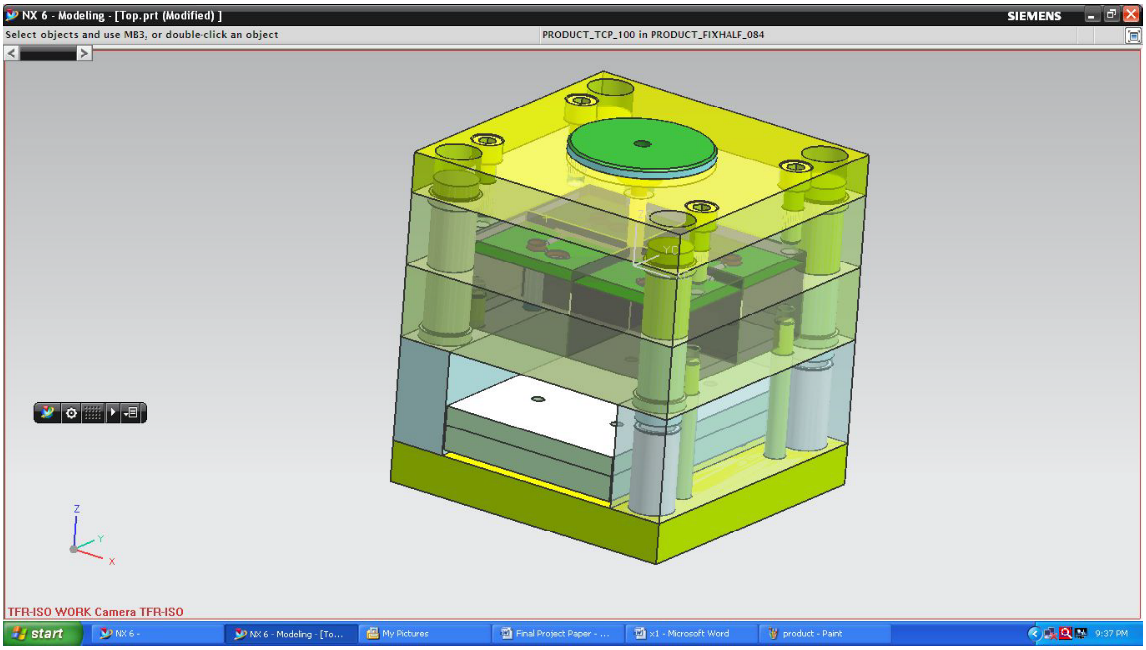
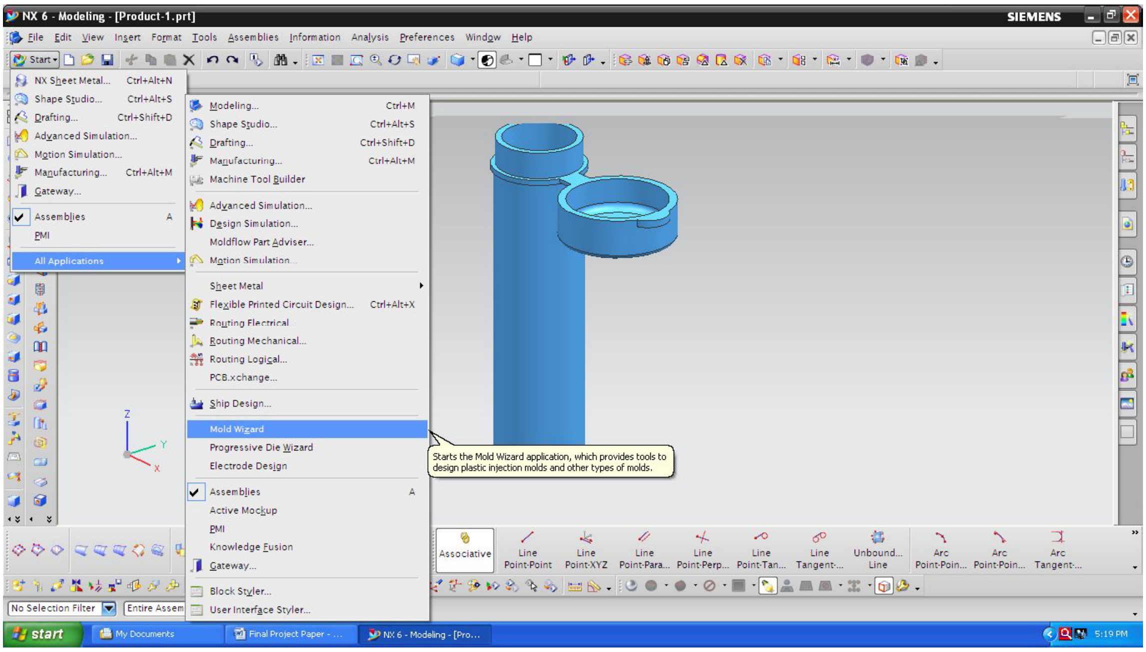
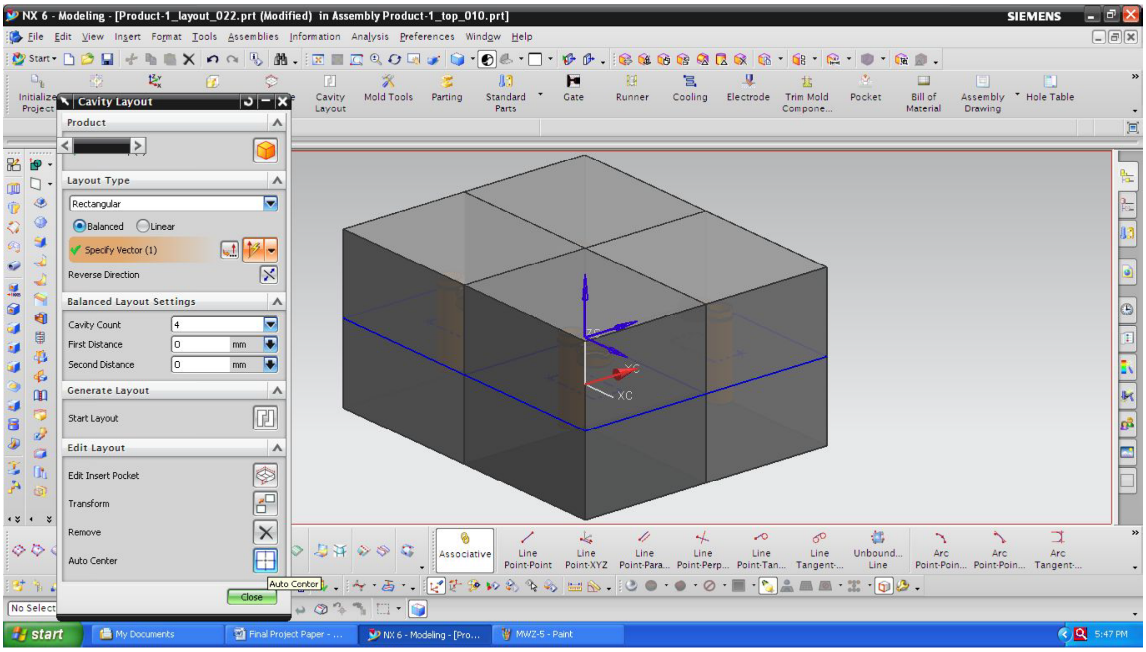



Figure5: Starting Mold Wizard.



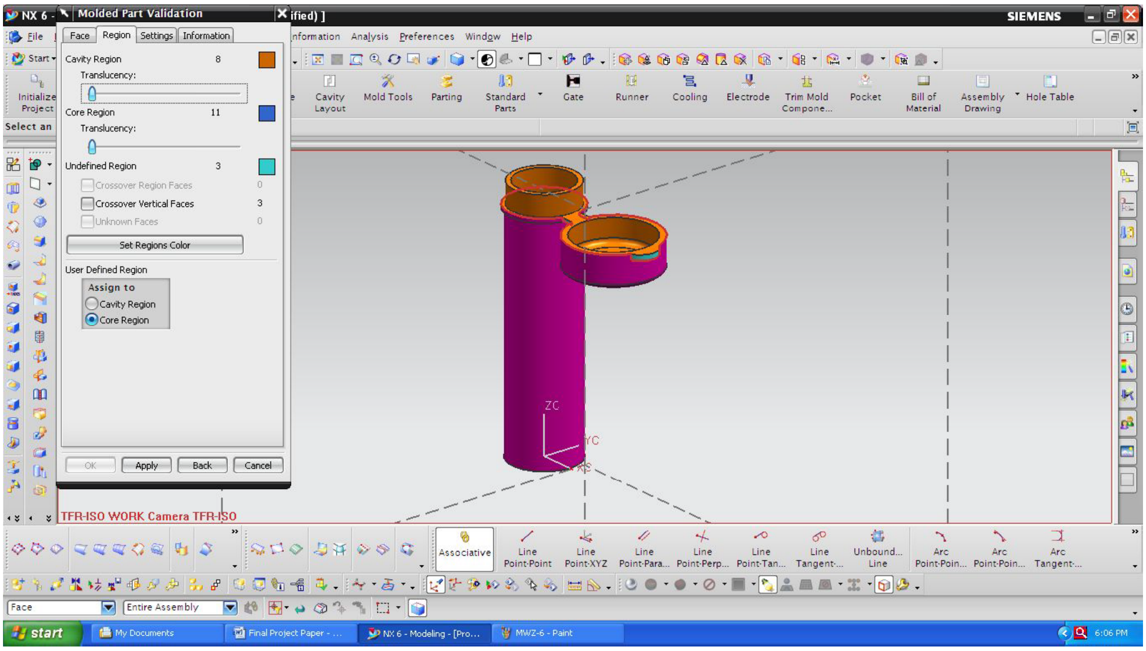

6. Select Cavity layout, rectangular, Cavity Count 4, Infer Vector YC, Edit layout, Auto center, Close. Figure 6.

Figure 6. Cavity layout selected Rectangular.



7. Select Parting, Design region, Keep existing, ok, region, cavity region, set region color, and apply, similarly for core. Figure 7.

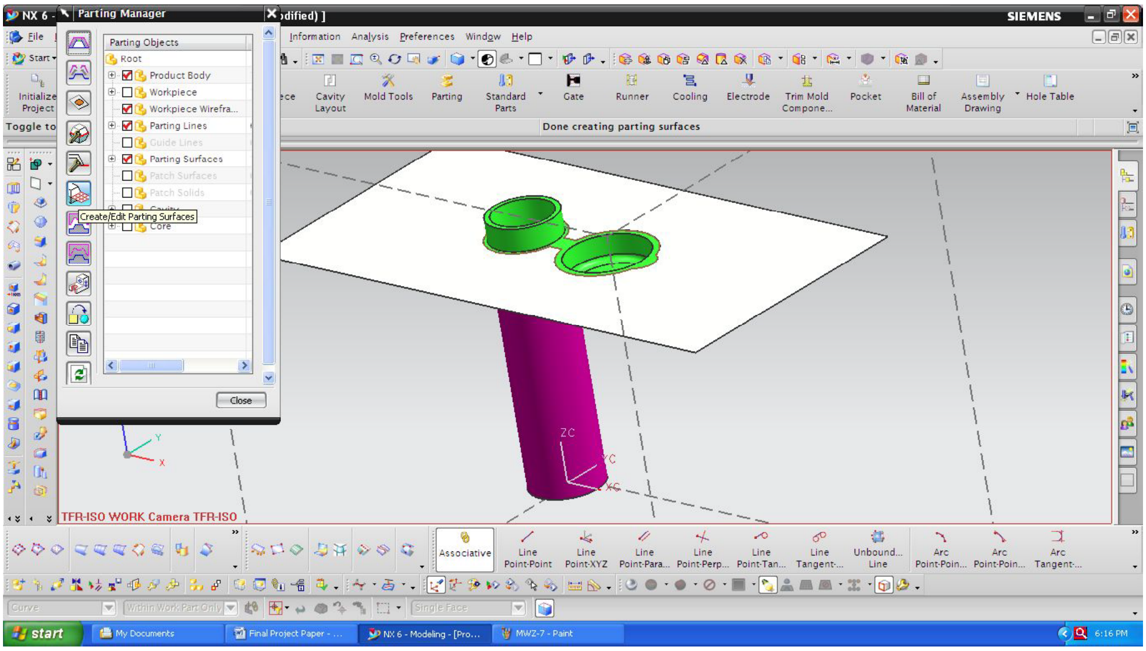
Figure 7. Setting different colors to surfaces extracted:  1. For core (upper) & for cavity (lower).



8. Select extract region & parting lines & give proper inputs, apply.

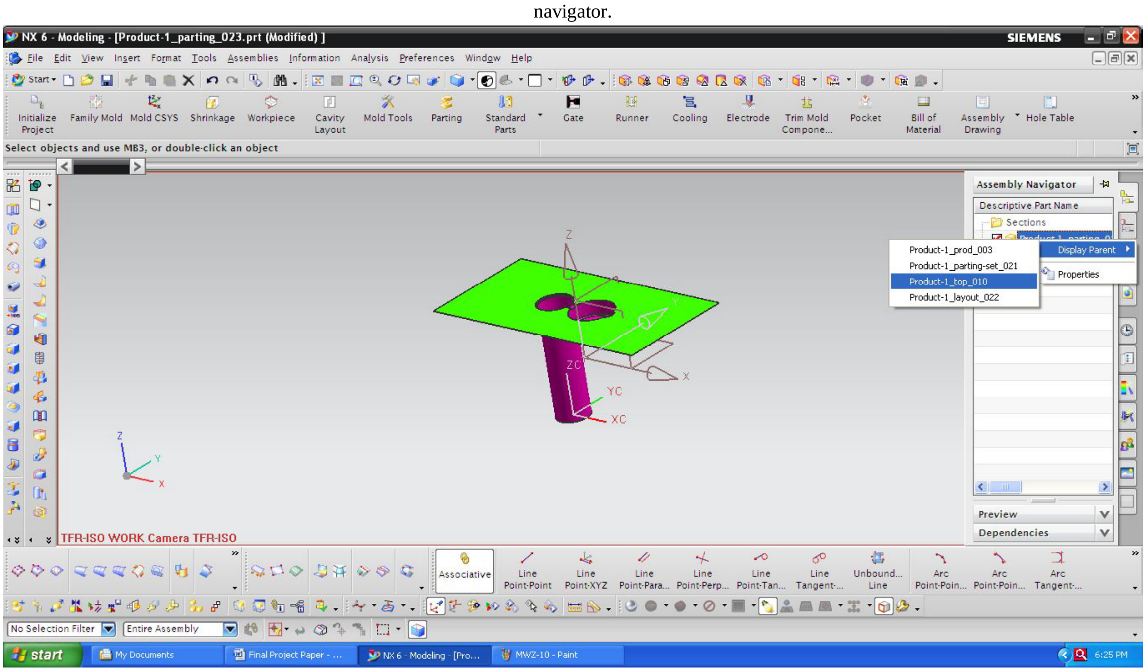
9. Select Edit parting surface, create parting surface, select bounded plane.  Figure 8.

Figure 8. Parting surface definition parameters.




10. Select cavity, core give proper inputs, select at right assembly navigator, , display parent product top. Figure 9.

Figure 9. Select option in assembly navigator.

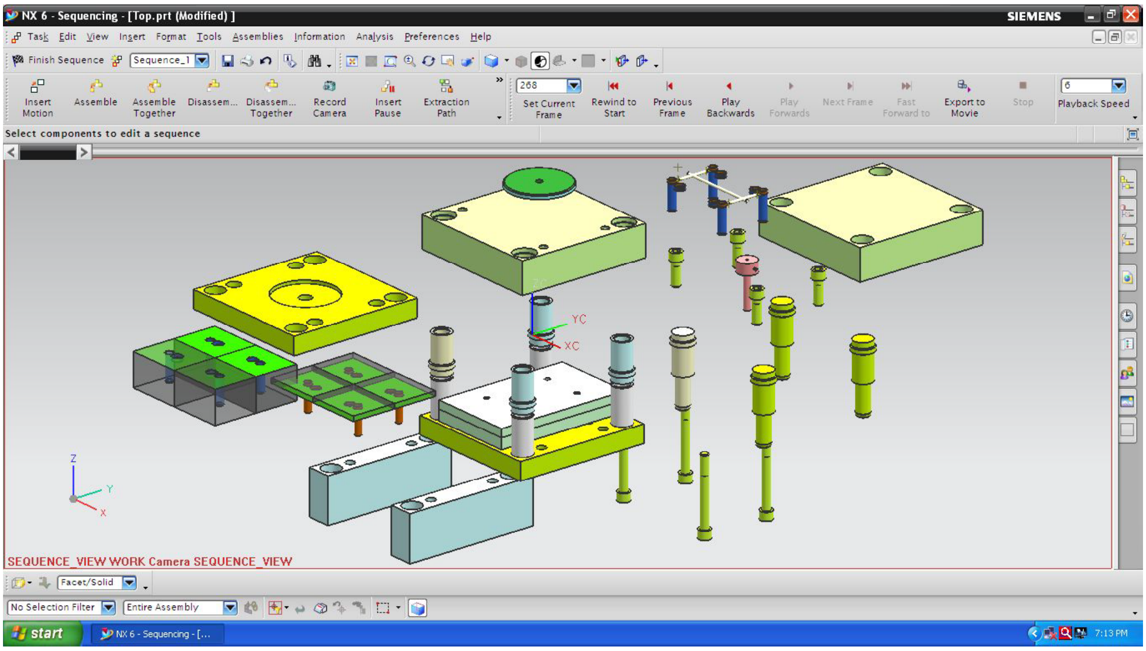


11. Select Runner in mold wizard menu; adjust A, B values, runner channel etc.

12. Select Gate option & input values.

13. Select mould base, cooling line, locating ring, sprue bush, pocket holes (standard parts) to add assembly parts of mold set. Figure 10.

Figure 10. Mold set parts disassembled.




VII.CONCLUSION

By using Mold Wizard facility in-built in Unigraphics, our design time for developing injection mold is considerably reduced. This reduces total lead time of manufacturing a plastic product, which helps to satisfy an urgent need of a modern customer. Secondly there is great increase in productivity if no. of cavities is increased with some additional cost of mold manufacturing, which remains a fixed cost. Thus use of a higher end cad software tool has become important in today's dynamic manufacturing environment from increasing machine productivity & labor productivity point of view.

VIII. ACKNOWLEDGEMENT:

The authors would like to thank directors of Thechnocad Solutions & Bhumi Tool room Solutions for significant information given on injection mold & various higher end cad softwares.

## REFERENCES

[1] Ivan Matin, Miodrag Hadzistevic, Janko Hodolic, Djordje Vukelic "AN INTERACTIVE CAD/CAE SYSTEM FOR MOLD DESIGN" Department for Production Engineering, Faculty of Technical Sciences, Trg Dostiteja Obradovica 6, Novi Sad, Serbia. 34th International Conference On Production Engineering 29-30 Sept 2011, University of Nis, Serbia.

[2] Nagahanumaiah, Bhallamudi Ravi, Narayan Prasad Mukherjee " A GENERIC FRAMEWORK FOR RAPID DEVELOPMENT OF INJECTION MOLDS AND PRESSURE DIE CASTING DIES" Scientist, Central Mechanical Engineering Research Institute, Durgapur, India Email: nagahanumaiah@hotmail.com,Associate Professor, Mechanical Engineering Department, IIT Bombay, India Email: bravi@me.iitb.ac.in ,Scientist, Central Mechanical Engineering Research Institute, Durgapur, IndiaEmail: npm50@hotmail.com.

[3] Weigang Hu and Syed Masood "An Intelligent Cavity Layout Design System for Injection Moulds" Industrial Research Institute Swinburne (IRIS), Swinburne University of Technology, Hawthorn, Melbourne, Australia 3122, International Journal of CAD/CAM Vol.2, No.1, pp. 69~75 (2002).

[4] Ž. Rutkauskas, A. Bargelis "Knowledge – based method for gate and cold runner definition in injection mold design" Ilmenau University of Technology, PF 10 05 65 98684 Ilmenau, Germany, Kaunas University of Technology, Kęstučio 27, 44312 Kaunas, Lithuania, ISSN 1392 - 1207. MECHANIKA. 2007. Nr4 (66).

[5] Suthep Butdee "Knowledge Engineering and Capitalization for Injection Mold Design" Integrated Manufacturing System Research Center (IMSRC) Department of Production Engineering, Faculty of EngineeringKing Mongkut's University of Technology North Bangkok (KMUTNB) 1518 Piboonsongkram Rd., Bangsue, Bangkok, Thailand 10800, 2011 3rd International Conference on Information and Financial EngineeringIPEDR vol.12 (2011) © (2011) IACSIT Press,Singapore.

[6] A B M Saifullah, S.H. Masood and Igor Sbarski "New Cooling Channel Design for Injection Molding" Proceedings of the World Congress on Engineering 2009 Vol I WCE 2009, July 1 - 3, 2009, London, U.K.

[7] Maria L.H. Low1 and K.S. Lee2 "3D RAPID REALIZATION OF INITIAL DESIGN FOR PLASTIC INJECTION MOULDS" Journal of The Institution of Engineers, Singapore Vol. 44 Issue 4 2004.

[8] B.Kayis, S. Kara "Set-Up Reduction in Injection Molding Process – A Case Study in Packaging Industry" The School of Mechanical and Manufacturing Engineering The University of New South Wales Sydney, 2052, Australia.

[9] A. Ghose, M. Montero, D. Odell "Characterization of an Injection Molding Process for Improved Part Quality" Berkeley Manufacturing Institute, Dept. of Mechanical Engineering University of California – Berkeley.

[10] Wong, C. T., Shamsuddin Sulaiman, Napsiah Ismail & A.M.S. Hamouda "Design and Simulation of Plastic Injection Moulding Process" © Universiti Putra Malaysia Press Department of Mechanical and Manufacturing Engineering Universiti Putra Malaysia, 43400 UPM, Serdang, Selangor, Malaysia, ISSN: 0128-7680 Pertanika J. Sci. & Techno!. Supplement 12(2): 85 - 99 (2004).