1998

Design of plastics injection molds for short-run production: a rapid prototyping approach

Christian Victor Signoret
Iowa State University

Recommended Citation

Signoret, Christian Victor, "Design of plastics injection molds for short-run production: a rapid prototyping approach " (1998). *Retrospective Theses and Dissertations*. 12381.
https://lib.dr.iastate.edu/rtd/12381

INFORMATION TO USERS

This manuscript has been reproduced from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps.

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6" x 9" black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.

Bell & Howell Information and Learning
300 North Zeeb Road, Ann Arbor, MI 48106-1346 USA
800-521-0600

UMI®

Design of plastics injection molds for short-run production:
A rapid prototyping approach

by

Christian Victor Signoret

A dissertation submitted to the graduate faculty
in partial fulfillment of the requirements for the degree of
DOCTOR OF PHILOSOPHY

Major: Mechanical Engineering
Major Professors: Jerry Lee Hall and Daniel X. Fang

Iowa State University
Ames, Iowa
1998

UMI Number: 9990508

Bell & Howell Information and Learning Company
300 North Zeeb Road
P.O. Box 1346
Ann Arbor, MI 48106-1346

Graduate College
Iowa State University

This is to certify that the Doctoral dissertation of

Christian Victor Signoret

has met the dissertation requirements of Iowa State University

Signature was redacted for privacy.

Committee Member

Signature was redacted for privacy.

Committee Member

Signature was redacted for privacy.

Committee Member

Signature was redacted for privacy.

Co-major Professor

Signature was redacted for privacy.

Co-major Professor

Signature was redacted for privacy.

For the Major Program

Signature was redacted for privacy.

For the Graduate College

DEDICATION

To my parents.

Gilbert and Liliane.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

AKNOWLEDGEMENTS

There are many people who helped me through this process and to whom I wish to express my appreciation. First, I want to express my gratitude to Dr. Jerry Lee Hall, my major professor and advisor. Throughout the course of my stay at Iowa State University, he gave me the freedom I wanted to select and work on my research, and I have always felt his support. His recommendations greatly helped me focus on the topic of this dissertation. He knew how to build up my confidence in my project when I needed it, and I have learned a lot from his advice.

I want also to express my thanks to Dr. Daniel X. Fang, not only for accepting being my co-major professor when I was already working on my research, but also for his continuous support and sincere interest in my research topic.

Another key person for making this dissertation possible is Lawrence Couture. His experience in machining and practical advice were invaluable for the completion of this thesis.

I am also thankful to the following institutions and foundations which made this possible:

- The *Consejo Nacional de Ciencia y Tecnología* [1], the Fulbright Foundation, the Institute of International Education and the College of Engineering at Iowa State University, for the scholarship I was granted,
- The *Instituto Tecnológico y de Estudios Superiores de Monterrey, Campus Querétaro* [2], for its financial support and for giving me a four-year leave of absence to continue my

---

[1] National Council for Science and Technology, Mexico.

[2] Monterrey Institute of Technology and Higher Studies (ITESM) at Queretaro, Mexico.

graduate studies, and

- The ASME Auxiliary, Inc., for selecting me for the 1996 Rice-Cullimore scholarship.

And last but not least, I am deeply thankful to my wife, Jenny, who encouraged me to undertake this project and has given me strength and love during these three and a half years, and to my children, Patrick, Philippe and Céline, for the joy they give me everyday.

1 INTRODUCTION

In the past ten years there has been a revolution in the field of design: The advent of the so-called Rapid Prototyping (RP) technologies, based mostly on additive processes, have allowed engineers to make physical models of their three-dimensional computer designs almost as easily as it is to print a two-dimensional CAD file.

As it can be seen in Figure 1.1 , based on Wholers' data [1], the sales of Rapid Prototyping equipment has had an exponential growth. Roughly a third of the almost 3,300 systems installed in the world were sold in 1997 only.

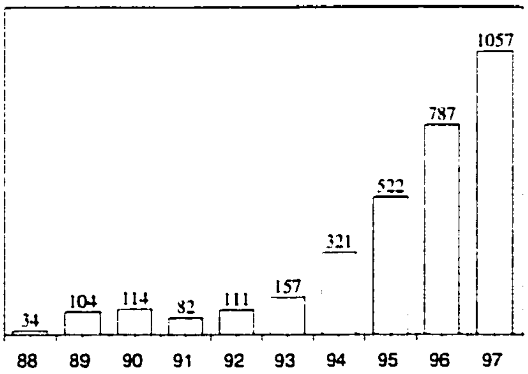


Figure 1.1 Rapid Prototyping Unit Sales Worldwide

More than one third of all the RP models are being used in industry as visual aids [2]. Marketing uses them to reduce the likelyhood of delivering an inadecuate product to the market. Engineers use them to verify form and function of the design and fit of parts in assemblies. The overall effect has been a reduction by up to 85% in cost and time-to-market [3].

Manufacturers immediately saw the great advantage of this technology to produce RP parts as patterns to make molds and tooling. RP can dramatically reduce the typically long lead time to produce cast parts and. at the same time. has come just in time to fill the void created by an aging. and now almost extinct. model-making profession.

Injection molders also are benefiting from this technology: lately. Research and Development (R&D) departments in both industry and universities have focused their attention on using RP as an instrument to develop prototype tools. This set of technologies is known as Rapid Tooling (RT). The technology developed in this dissertation. can be included in this category since it presents an RP approach in the design of plastics injection molds for short-run production.

Research goals

The main objective of the present dissertation is to present a Rapid Prototyping approach in designing plastics injection molds for short-run production. The specific goals are to:

- Present the current state-of-the-art in design and manufacture of molds for plastics injection for short-run production. focusing on the most important systems commercially available for Rapid Prototyping and Rapid Tooling.
- Present several of the current research projects being performed in Rapid Prototyping and Rapid Tooling and show how they can potentially be applied to make short-run plastics injection molds.
- Present specific considerations. needs and requirements for the design of plastics injection molds for short-run production.
- Define a set of rules and a methodology to determine the best technology to make a specific plastics injection mold for short-run production.
- Design. build and test a standard mold base especially conceived to hold short-run production molds.
- Introduce an innovative way to make mold cavities and mold cores for plastics injection by combining "conventional" CNC milling and direct casting of inserts in the mold plate. (Theory of the process and experimental results)

Need for the research

In the past two decades there has been a clear new trend in industry, first, "to do more with less" and then, "to do less with less". To accomplish this, industry has focused its energy on reducing the time-to-market necessary to design products. Usually the high cost of designing a new product is associated with the time spent at every step of the process. New technologies and strategies have been implemented to shrink this time. These technologies are often called "Time-Compression Technologies" (TCT) [4]. Of course, time cannot be compressed, but it can be used more wisely by using these new technologies. Design and manufacturing are not anymore seen as independent processes but as part of an integrated system that also includes other areas such as marketing, management and information systems.

To do less with less might seem incongruent with the population and economic growth the world is experiencing now, but it actually has to do with these factors and is associated with the relatively recent advancements in communications and computer technologies.

Design engineers nowadays need to think of how to dispose of or how to recycle the product they are designing from the very moment they start designing it [5]. The lifetime of a product is shrinking more and more; as technology advances, many products become obsolete sometimes in a matter of months. That is the main reason that relatively short production runs are needed. There is also a need for quickly testing new products, and low-cost prototype runs are required before committing resources for short and medium production runs.

Furthermore, globalization of the economy and free markets have also put pressure on the demand for fewer, high quality products since more competitors are sharing the same market niche.

To respond to these pressures, the manufacturing sector of industry has been looking at technologies to reduce the time-to-market. Among others, technologies such as Concurrent Engineering (CE), Just-In-Time (JIT), Theory of Constraints (TC), Pull systems, Kanban, Quality Function Deployment (QFD), Quality Engineering, Taguchi methods, Computer Aided Design (CAD), Computer Aided Manufacturing (CAM), Flexible Manufacturing Systems (FMS) and Computer Integrated Manufacturing (CIM) have been developed and adapted to the current requirements.

In particular, the concepts of Concurrent Engineering and Just-In-Time manufacturing require machines capable of prototype and short-run production and quick set-up times.

In the machining environment, the response to this need has been filled by flexible manufacturing cells that include among others: Computer Numerically Controlled (CNC) machines, robots, automated guided vehicles (AGVs) and devices such as conveyor belts , modular/flexible and automatic fixturing devices controlled by programmable logic controllers (PLCs) all of this centrally controlled and monitored by stand-alone or networked computers. But other manufacturing processes have not yet gotten to that level of flexibility as it is the case of casting and molding.

In the plastics injection molding environment, in particular, there is a very strong need to have a flexible process. Most machines and devices nowadays contain plastic parts that have been produced by this process and there is no doubt that it will continue to be favored and that more applications will be found as new developments in plastics and composites occur. But this process, as it is currently known, is prohibitively expensive for the short-run productions needed in today's industry.

To make this process more flexible, there is a need to analyze and improve the two main components of the process: the molding machine, and the mold.

The new injection molding machines have become more flexible in the sense that now most of the parameters of the machines can be controlled by a computer. In this way, shot volume, injection pressure, clamping force, holding pressure, cycle time, etc., can be programmed and electronically stored for each individual part. Some machines already have digital interface capabilities to connect them to devices for automatic loading/unloading of the molds and inserts or to communicate with other peripherals.

Standardization of some mold elements have given some flexibility to the fabrication of the mold itself: standard mold bases, sprues, ejector pins, plates, etc. have reduced the overall cost of making molds and have improved greatly the lead time to make them but the mold itself -the actual cavity and core set- still remains as the single most "rigid" component of the system.

The "rigidity" of the mold comes mainly from the lead time to make it and the high cost associated with it and from the fact that once a mold has been made, it cannot be easily and quickly changed. So, in order to highly improve the flexibility of the injection molding process, there is a need to research new materials and new ways to make molds for short-run production and for prototyping.

In this dissertation, the focus of the research will be on designing new molds combining the new capabilities of tridimensional solid modeling software, CNC machining, casting techniques, and the mold bases required for these molds.

Thesis organization

The thesis organization is fundamentally based on the objectives to be attained:

- In Chapter 2, there is a literature review of the current technology to fabricate prototypes and short-run production molds, focusing on the so-called Rapid Tooling technologies which are based mostly on Rapid Prototyping technologies.

- A study of the special considerations that have to be taken when designing prototype molds and/or short-run production molds. A set of design rules and a methodology to determine the best technology to make them is presented in Chapter 3.

- In Chapter 4, the design process of a special mold base to hold short-run production molds -based on the considerations cited in Chapter 2- is shown. This mold base was manufactured and tested, and specific conclusions and recommendations are made.

- In Chapter 5, the design and manufacturing process of a mold with removable cores is shown. In this case, the product to be molded was redesigned in order to simplify the mold design and speed the "conventional" CNC machining.

- In Chapter 6, an innovative Rapid Tooling process in which simple parts of the mold are machined and hard details are reproduced by directly casting an insert into the mold is presented. A specific experimental study case illustrates the process.

- Finally, the general conclusions of this research and recommendations for further research are summarized in Chapter 7.

2 LITERATURE REVIEW

The fields of Rapid Prototyping (RP) and Rapid Tooling (RT) are changing at a fast pace. With the advent of new technologies, the definitions of these terms are changing also. In this chapter, a review of the main current commercial and experimental systems is done.

Rapid Prototyping

Background

Traditionally there are three general manufacturing or fabrication processes, manual or automated, to produce a solid object:

- **Subtractive processes.** These processes start from a solid block of material bigger than the desired object. Material will be removed until it is shaped to the desired geometry.
- **Additive processes.** Successive parts are "added" and combined to make the final object.
- **Formative processes.** These processes, which are also called "net-shape" processes, take some material and shape it by mechanical forces to a desired geometry. These processes include bending, stamping, coining, and molding of melted and curable materials.

There are also processes that combine any of the basic processes described above, to optimize the fabrication process.

Definition of Rapid Prototyping

Rapid Prototyping is defined by Burns [6], as a process in which:

- The process takes a shapeless material (blocks, sheets, liquid) and transform it in a solid object with a definitive shape.
- The process is done without any human intervention.
- The shapes produced by the system may include any three-dimensional geometrical complexity.
- No tooling is required to make different shapes.
- Each item produced is a single object. No assembling is needed.

Although the term Rapid Prototyping (RP) is definitively the most common, other terms are used for referring to similar technologies [7]. These names include: Stereolithography (SL), Three-dimensional printing (sometimes wrongly called "3D Printing", which is a copyrighted commercial process), Solid Freeform Fabrication (SFF), Solid Freeform Manufacturing (SFM), Automated Fabrication, Layered Manufacturing, Desktop Manufacturing, Direct CAD Manufacturing, Instant Manufacturing, Layer Manufacturing, Material Deposition Manufacturing (MDM), Material Addition Manufacturing, Material Incress Manufacturing. Each term either implies a particular technology, or is intentionally independent of the fabrication process.

Historical perspective

The roots of RP can be traced to at least two areas: topography and photosculpture [8]. At the end of the 19th century, Blanther suggested a layered method for making topographical relief maps. In his method, he stacked wax plates cut following the contour lines of topographical maps. His technique was improved or modified by others, with the advent of new technologies and materials.

Photosculpture arose also in the 19th century with the intent to replicate three-dimensional objects. In 1860, Willème had a photosculpting studio in Paris. It was a circular room, with 24 cameras placed equally about the circumference of the room. The object, or person was placed in the center of the room. The silhouette of each photograph was then used to carve out 1/24th of a cylindrical portion of the object.

Commercial additive RP systems

Some of the most common additive Rapid Prototyping systems are described below. Most other commercial systems are similar and will be named later in this section.

Stereolithography apparatus (SLA). In this process. solid models are produced from liquid photopolymer. A laser generated ultraviolet light is focused at the liquid surface. Controlled to scan the surface. the spot traces the desired shape of a layer. The surface is recoated. and the process repeated. until the part is finished. This is the first. and the most common system currently in use.

Solid Ground Curing (SGC). It is also known as the Solider process. This process also utilizes a photosensitive polymer. but the entire coated surface is cured instantaneously by a burst of ultraviolet light. The desired shape is defined by an electrostatic mask on a clear glass. acting as a stencil. The mask is cleared and a new one is produced for the next layer. After each layer. the excess polymer is removed. the empty spaces filled with wax. and the top surface milled to control its height.

Selective Laser Sintering (SLS). This process is similar to SLA but in this case. instead of a liquid photopolymer. powders are used. There are currently different types of powders. from filled and unfilled thermoplastic polymers. to elastomers. to ceramics and stainless steels. In all cases. these powders are sintered by a laser focused on the top surface. The unused powder is left to support the part.

Laminated Object Manufacturing (LOM). In this system. sheet material. coated with a heat-sensitive adhesive is used to make each layer. The current layer is glued to the previous one by a hot roller. The contour of the shape is cut by a laser beam. The unused part is laser-cut in grid pattern to facilitate the extraction when the part is finished. The original system used paper as the sheet material. and yielded plywood-like parts. New materials are available now. including several thermoplastics. ceramic powders and metallic powders. The latter require post-processing furnace operations to sinter the powders.

Fused Deposition Modeling (FDM). In this process. models are made from extruded thermoplastic. The system is similar to an X–Y plotter. but with an extrusion head. instead of a pen. This deposits a series of extremely thin layers of material to build the part. The process is fast but accuracy is relatively low, compared to SLA, SGC, or SLS. It is often considered more an "office modeler" than a Rapid Prototyping machine.

Multi-Jet Modeling (MJM). This is an inkjet-based system 96 "print" heads that builds finely detailed solid objects using a thermopolymer material. The "print head" goes back and forth just like a printer. For part geometries wider than the print head, a cross-axis is provided to reposition the part under the print head as necessary. As the FDM system, this system is considered an "office" modeler.

Three-Dimensional Printing (3DP). In this process, licenced to Z Corp., the models are made by "printing" a binder on a powder. This process, currently the fastest, utilizes a cellulose based powder material. It produces a green part, which can be infiltrated with wax or epoxy to improve its strength properties. It is considered an "office" modeler.

Classification of commercial Rapid Prototyping systems

Table 2.1 defines and classifies the commercial additive fabricators by the technology used by the system [9].

Rapid Tooling

Rapid Tooling (RT) is a term used to define any technology that uses some RP technology to produce different kinds of tools. Few of these technologies are almost completely automated, most are not. The current term RT does not imply if the tooling is for short or long run production. Although it is not specifically for plastics injection molding, most of the RT being researched and developed nowadays is in that field.

Rapid Tooling history

It takes months to produce injection molds. As the required number of parts to be produced from one mold decreased, it became a necessity to lower the cost of the mold. Pressure to shorten the lead time was also increasing. Typically a mold impression would be made with

Table 2.1 Commercial additive fabricators

PROCESS	LINE	VENDOR	HEADQUARTERS
Selective curing	SLA	3D Systems	California, USA
(All laser-based	SOUP	CMET	Japan
except the Solider)	Stereos	EOS	Germany
	SLP	Denken	Japan
	JSC/SCS	D-MEC (Sony)	Japan
	Solider	Cubital	Israel
	Soliform	Teijin Seiki	Japan
	Meiko	Meiko	Japan
	LMS	Fockele and Schwarze	Germany
	UniRapid	Ushio	Japan
	COLAMM	Mitsui	Japan
	Solid Imager	Aaroflex	Virginia, USA
Pattern lamination	LOM	Helisys	California, USA
	Solid Center	Kira	Japan
	RPS	Kinergy	Singapore
Selective sintering	Sinterstation	DTM	Texas, USA
	EOSint	EOS	Germany
Continuous deposition	FDM	Stratasys	Minnesota, USA
Drop-on-powder	DSP	Soligen	California, USA
deposition	Z	Z Corporation	Massachusetts, USA
Drop-on-drop	Model Maker	Sanders	New Hampshire, USA
deposition	Actua MJM	3D Systems	California, USA
	Modeler	BPM	South Carolina, USA

an EDM machine. The EDM electrode is usually made of graphite or copper, and is made with a CNC machine. These electrodes are also finished by hand. Although an excellent and proven process, it is too long for today's requirements.

CNC can be said to be the first approach to Rapid Tooling. As computer software to generate CNC code had more capabilities and machines had more memory, complex three-dimensional machining became a reality. Ball-end milling, required for that, is however a lengthy process. Higher milling speeds were only possible by changing to more machinable materials. Aluminum alloys became the top choice for CNC milled molds. Although aluminum molds were only considered prototype molds by some manufacturers, some others found that they could meet the quality and time requirements of their clients with them.

Another approach to Rapid Tooling has been to reduce the time by making the EDM electrode using the new RP technologies being developed. Gupta and Hall [10] describe such an approach for die manufacturing. Other approaches have being developed and commercialized. and focus on making the mold inserts directly or indirectly with RP technology.

Classification of Rapid Tooling

There is an informal classification of Rapid Tooling techniques. based on the type of tool made [11]. The first one is defined as **Rapid Soft Tooling**. and refers to the fabrication of a silicone RTV (room temperature vulcanizing) rubber molds. This process utilizes a master in the "positive" form of the final part. Silicone is cast in a box where the master is suspended. Then. the un-cured RTV is degassed in a vaccum chamber. and placed in a controlled-temperature oven (50 degrees Celcius) for the curing process. The cured RTV mold is cut with a scalpel. and the master removed. Any two-part polyurethanes can be vaccum poured into the mold. These urethane resins are available in a wide range of properties to "simulate" a wide range of thermoplastics.

At the other end of the spectrum. there is **Rapid Hard Tooling**. Rapid Hard Tooling utilizes RP technologies to produce metal mold inserts. These mold inserts can withstand thousands. and perhaps millions of shots. There are several different technologies: "Keltool" and "RapidTool". are two of them and are described later.

In between these two categories. there is **Rapid Bridge Tooling**. Rapid Bridge Tooling produces injection molded parts in the final production material. The direct rapid manufacturing of the tool is one approach. The indirect rapid manufacturing is the other. In the first case. the whole tool (mold insert) is made in an RP machine. 3D Systems calls this process DAIM (direct ACES injection molding). ACES stands for Accurate Clear Epoxy Solid. and refers to the epoxy and build type used in their SLA (stereolithography apparatus). Another approach is to fabricate just a shell with the RP machine (to save time and material) and backing it with AFE (aluminum filled epoxy). This approach has the advantage to speed up the molding process cycle time. because of the increased thermal transfer of the tool. Its main disadvantage with respect to DAIM is that it is not completely automated anymore. In the indirect approach. the part model-pattern is used to make the mold impression. Several of these techniques involve making first a hard shell on the pattern. and then backing it with different materials.

The commercial Rapid Tooling systems

In the **3D Keltool** process. originally created by 3M Co.. a stereolitography part is made as a negative of the final shape. The negative pattern is placed in a box filled with RTV rubber which cures. and hardens. Ultra-fine metal powder (Stellite. A6 tool steel. or copper-tungsten) is poured into the rubber positive. The powder. which has a thermoplastic binder. has a varied size distribution for tight packing and high fill ratio. The molded powder form is heated at 100 degrees Celcius to for a green part. The green part is then demolded and fired in a furnace. at 1300 degrees Celcius. The binder burns and the part is sintered. The sintered part can be infiltrated with copper to produce a 98% density tool. Resulting small parts are reported to have accuracies in the order of 0.001 inch per linear inch. Shrinkage from the original CAD model is claimed to be under 0.010 in/in.

The **Rapid Prototype Composite Mold** developed by Albright Technologies. Inc. [12] starts with a positive pattern generated by some RP machine (SLA. LOM. SLS. CNC machined metal plastic). This pattern must be built with the desired surface finish and dimensional compensation for the molded resin shrinkage. Aluminum mold cavities are then rough machined. to a high tolerance contour of the model. The pattern is then placed in the cavity and a specially treated rubber is cast to reproduce the fine details of the pattern. Dimensions of tight tolerance can be treated by the incorporation of metal inserts.

EOSINT M was the first commercial system for direct laser-sintering of metallic powder [13]. The main application is in toolmaking for injection molding and related production methods. This layer manufacturing technique enables metal parts to be built directly from CAD data. A thin layer of metallic powder is spread over a building platform. and locally sintered by a laser beam. Layer by layer the three-dimensional geometry is reconstructed in every detail.

The **Nickel Ceramic Composite (NCC)** tooling system, developed by Cemcom. Corp. [14], starts from an RP generated model. The nickel is electroformed over the tool model. The nickel shell is backed and bonded to a standard mold base with chemically bonded ceramics (CBC). Then the tool model is extracted, ejector pins drilled, runners machined, etc. A typical NCC mold without a cooling system has a cycle time 20% longer than a steel mold. It is designed for making 10,000 to 50,000 parts.

Little is publicly known about the **Prototype Hard and Soft Tooling (PHAST)** process developed by Tobin and licenced to Plynetics Express [15]. The process is said to coat a ceramic slurry on an RP pattern, and it is then backed with steel powder (50% steel and 50% air). Copper is infiltrated into the porous material in a furnace cycle.

Polysteel, a process developed by Vawter of Dynamic Tooling, uses an RP model as a pattern to produce mold inserts that are 90% steel and 10% epoxy. The resulting molds are several times stronger than aluminum and significantly harder. They are ideal for prototyping glass filled resins. Because of its good thermal conductivity, injection molding cycles are similar to solid steel molds. Current lead times for molds and molded parts is 10 to 15 days.

DTM's **RapidTool** process has improved lately. In **RapidSteel** version 2.0, stainless steel powder is selectively sintered in an SLS machine. The laser-sintered green mold inserts can go directly from the SLS machine to the furnace. The part is infiltrated with bronze. The mold can last for 150,000 plastic parts, and hundreds of aluminum, zinc, or magnesium parts.

DTM's other RT technology is the **Copper Polyamide RT**. It allows for short-run production (several hundred parts). The powder used in their SLS machine is a mixture of copper and nylon. No post process in a furnace is necessary. The composite tools obtained are machinable.

ExpressTool is a firm working on two RT technologies. The first one based on an electroforming process which is already available. The second one is still in development and little is publicly known. It is described as a powder-metal technology that produces hard tooling of chromium-carbide. The electroforming process is being co-developed with Hasbro, Inc. It produces a 1- to 2-mm-thick layers of nickel on a CNC machined graphite mandrel. The nickel shell is backed with aluminum filled epoxy. This process can produce large parts.

The **Prometal Rapid Tooling System** is an SFF machine that creates steel molds. It is based on the three-dimensional printing (3DP) technology developed by Sachs at the Massachusetts Institute of Technology. It uses an electrostatic ink-jet printing head to selectively deposit a liquid binder onto a powder. This process goes on layer by layer and yields a "green" metal parts that is sintered and infiltrated with a secondary metal.

Optomec Design. CO. has licenced the **Laser Engineered Net Shaping (LENS)** technology developed at the Sandia National Laboratory. in Albuquerque. New Mexico. The LENS process makes fully dense objects by injecting metal powder into a pool of molten metal created by a laser beam focused on a substrate.

A similar method. the **Direct Metal Deposition (DMD)**. has been used to create H-13 tool steel components. With this method. injection molding dies and trimming dies have produced. These components had dimensional tolerances within a few hundredths of a millimeter [16. 17].

Experimental Rapid Tooling systems

Rapid Tooling based on Rapid Modelers. Several new RT systems are being researched and developed. At the Rapid Prototyping and Manufacturing Institute (RPMI). at Georgia Tech. research is being done on epoxy tooling made from master patterns fabricated by the 3D Systems' Actua Multi-Jet-Modeler (MJM) [18]. The MJM makes paraffin wax parts. Because the epoxy can be directly applied to it. it speeds up greatly the process of making prototype molds.

New tooling materials for CNC machining. These composite materials are very tough. and their machinability makes them good candidates for prototype tooling. Filipiak and Kotnis [19]. report on a case study done with a composite board developed by Prince/Ciba and high-speed machining. The results are compared with other traditional processes and materials. It shows that it is possible to make inserts in 15–20% of the time required with traditional methods and materials.

Hybrid systems. Several "additive" RP methods are actually adding material and then cutting it, as it is the case in LOM. Many others also mill the top surface of each layer to control the "Z" axis dimensions. There are actually some real hybrid systems being developed and tested. This is the case of **Shape Deposition Manufacturing (SDM)**. SDM is an SFF method that has the capability to directly create functional metal shapes which are dense, metallurgically bonded, geometrically accurate, and with good surface appearance [20, 21]. To form each layer, the material is deposited as a near-net shape using a novel weld-based deposition system called microcasting. Then the material is net shaped with a 5-axis CNC milling machine. Finally, the part is transferred to a stress-relief station, such as shot-peening, to control the residual stresses buildup. In this form, the construction material and the support material are alternatively deposited and shaped.

Combining additive RP methods with CNC routers. A unique combination of SLA and CNC router milling has being used by ARRK Creative Network Corp. [22, 23] to produce quality injection molding prototypes. In this process, the "rough" parts are fabricated by the SLA and the fine details are routed out. It combines the best qualities of these additive and subtractive systems.

Experimental applications of Rapid Tooling

Several new applications are being developed for the existing RT systems. Initial results of simple **Ceramic Injection Molding** in SLA solid epoxy molds have been successful. This technology, derived from plastic injection molding, requires low pressure and low temperature to inject the ceramic/wax slurry. The ability to predict the shrinkage of the sintered ceramic parts still remains a challenge. Small corrections can be made in the 3D computer model of the mold and in a matter of hours an SLA machine can rebuild a new mold [24].

Additive versus subtractive methods

The traditional subtractive methods used in Rapid Prototyping and Rapid Tooling have been compared to the new additive methods. Both methods have their supporters. The general trend is to think that the additive RP methods will replace the subtractive ones. Subtractive RP supporters point out that additive methods are relatively inaccurate and that long flat parts tend to sag, warp and curl [25]. New improvements in CNC machining, and CAM software supporting it, are closing the gap in this "competition" to produce Rapid Tooling [26]. Several case studies reported by Smith [2], show that different applications require different methods. It is the author's belief that the future systems will combine more and more both approaches. They are not mutually exclusive, and actually many of these "additive" systems are in fact hybrid systems.

Typically, additive systems are faster to produce a part and most case studies have shown that. There are however some other cases where this is not true. As reported by Song, Park and Ha [27], Rapid Tooling methods (LOM combined with casting) is still not comparable to five-axis milling with high-speed spindle. In their study, they manufactured a five-blade ship propeller using both methods and found out that for roughly the same cost, five-axis milling was faster and more accurate than using LOM combined with sand casting. The conventional method basically took a little more than a week, while the Rapid Tooling method took a little more than two weeks.

3 MOLD DESIGN RULES FOR SHORT-RUN PRODUCTION

The definition of short-run production (SRP) in plastics injection molding varies from one person to another, and from one company to another. There is actually no firm definition of what a short-run production is. For people in industry used to millions of parts, tens of thousands of parts is an SRP; for others, a few hundreds means SRP, and for others just a few tens can be defined as SRP. The mold quality required for a specific part is a function of the expected life, the accuracy and surface finish of the part, and the abrasiveness and chemical composition of the thermoplastic to be injected. In the next section, a tool classification based on these variables is presented.

Tool classification

Morgan Industries, Inc., a company that manufactures injection molding presses for prototype and short-run molding of thermoplastics, defines three types of molds, or tools, for injection molding [28]: Automatic molds, semi-automatic molds and hand molds. These molds can be classified in three categories: Class A, class B, and class C.

Class A tooling

Class A tooling is built for an extended life, high throughput, and speed of operation. This kind of tooling is intended only for very large productions, to justify the high cost of the mold. Usually it has several cavities. It is made with the finest materials, the surfaces are treated, and the accuracy and tolerances are very tight. They have all the peripherals that can improve the cycle and throughput of the molding process: Hot-runners, temperature control, ejector systems, sliding inserts, etc., for each particular case. To produce class A tooling, experienced designers and toolmakers are required. It takes several weeks, sometimes months, to have the final tool. Parts molded in this type of tooling rarely require any secondary finishing operation.

Class B tooling

Class B tooling typically has more informal detail. It is made with more common materials, that are easier to machine, and it is usually intended for semi-automatic production. Parts made with this kind of tooling may require secondary finishing operations. Cross-holes and undercuts are produced by hand-pulled or loose cores. Internal and external threaded cores are rotated manually to remove them. The surface finish of the mold may include engraving, texturing, polishing or plating. The mold may have more than one cavity. A class B mold is an excellent, low-cost tool for medium and short-run production.

Class C tooling

Class C tooling includes most hand molds: it is intended for very short-run production and therefore its cost must be as low as possible. It just has to last for the intended number of parts required. To make it, any low-cost and easy-to-work materials are acceptable as long as they can withstand the temperature and pressure of the process. Usually the molded parts do not require high precision, and tolerances are large. Parts usually require secondary operations to avoid having to make an expensive detail in the mold. The objective is to accomplish the production of the finished parts at the lowest cost possible. Every tooling decision must be made with economics in mind. This usually leads to reduce the tool building time, and to increase the molding and finishing time. According to this classification, class C molds -and sometimes class B molds- will be the focus of this dissertation.

Class characteristics

More characteristics related to the classification of molds can be seen in Table 3.1. Note that the costs shown in that table are relative costs since they will vary according to the value of the polymer used at some time.

Table 3.1 Comparative characteristics of tool classes

Part	Class C tool	Class B tool	Multi-cavity class B tool
Part design firm and proven	No	Yes	Yes
Part material firm and proven	No	Yes	Yes
Post finishing required or acceptable	Yes	Slight	Minimal
Quantity requirements - Total	To 500	To 5.000	To 25.000
Finish requirements - As molded	Open	Good	Good
Part cost allowance	Over 50c	Over 25c	Under 25c
Toolmaking skill available	Slight	Moderate	Moderate

Design considerations for short-run production tooling

Molds for short-run production will follow the same basic design rules for standard injection molds. Most of these rules and considerations can be found in manufacturing textbooks and the intent of this section is to discuss these considerations for a Short-Run Production approach and to define specific rules to design SRP molds in the following section.

A simple analysis of the costs involved in an SRP mold will define some parameters in mold design that can lower the total cost of the production.

Cost considerations in SRP Injection Molding

The cost of one finished molded part is basically a function of four costs:

Mold cost = the cost of the mold

Process cost = the cost of the molding process

Finish cost = the cost of the secondary finishing operations

Material cost = the cost of the material

In very large production injection molding. all the efforts are focused on reducing the cost of the material. the cost of the molding process and the cost of the finishing process by improving the mold. Contrary to that. in SRP all the efforts are focused on saving money on the mold by making it faster. making it simpler. with less details.

The cost of the material is basically independent and constant: Only a very slight saving in material can be achieved by having a better mold. However, the more sophisticated -and expensive- is a mold, the more savings we will have in the molding process cost and the finishing process cost, and vice versa.

When designing an SRP injection mold, this fact must be remembered, and a cost analysis must be done. This analysis must be simple and fast since the time factor in SRP injection molding is often very important. An extensive and complex analysis would probably be more expensive than the savings it could eventually achieve. Most of the decisions to reduce the cost of a mold are related to the elimination, or modification, of details of the part to mold. These details will eventually have to be done as a secondary operation or, if it is acceptable, will remain as they were modified.

As an example, if a simple straight hole is required on the object and the number of parts is very low, usually a secondary drilling operation, using a simple modular/flexible fixture to locate the part, is the best solution, and the total cost of that SRP is lower. But if the number of parts is relatively large, an insert in the mold would be the best solution. Many of these decisions must also be a function of the manufacturing facilities available.

To aid in this decision making process a simple computer program or a computer spreadsheet can be used. In Appendix A there is an example showing the use a simple computer spreadsheet. The cost analysis program can be as simple or as complex as the user wants it to be, and can be customized very easily. It is only a tool to define the approximate cost of a single molded part. Good results are highly dependent on the accuracy of the data entered. The accuracy of the data depends significantly on the experience of the person who estimates the implications -in time and money- of simplifying or eliminating details in a mold.

General principles to reduce the mold cost

Principle 1: Changing the mold material. By changing to a very machinable material, the cost of machining decreases as it can be done at very high speeds, minimizing tool wear and improving the surface finish so that almost no polishing is needed. On the other hand. by doing this. the mold also might wear more easily, especially with abrasive materials such as glass fiber reinforced polyamides. In these cases. an experimental study of the abrasiveness of certain materials is needed in order to assure that the mold will last for at least the minimum number of parts required. If this study is not available. a prototype mold can be made and production started. A statistical process control will then be needed to determine if. and when. another mold is needed to finish the required production. Sometimes this pragmatic approach might be even less expensive than machining a "better" material to make the mold.

Principle 2: Machining the mold in one operation. The idea behind this is to have the mold material mounted on a fixture device in such a way that all important mold surfaces will be generated during the machining without disassembling the fixture and preferably with the same tool. This approach is one of the most important -if not the most important- to obtain a good mold. with good geometric and dimensional tolerances. in a very short time. A common quality problem in low-cost molds is the misalignment of the mold halves and parallelism problems at the parting line planes: these problems will lead to offset halves. flash at the parting. and frequently short-shots. By using this simple rule. all these problems can be avoided. To accomplish this. a good and simple modular fixturing system is a must. This topic on flexible and modular fixturing is so important for the manufacture of the mold. and also for its assembly on the injection molder that it will have its own section on this chapter.

Principle 3: Redesigning the part for SRP injection molding. It is faster and less expensive to machine a simple mold. This obvious truth has many implications in the way a part has to be designed. or redesigned. Simplifying a mold means that the part has to be redesigned according to some rules that are described later in this chapter. The basic idea behind this rule is to maximize the efficiency of the machining of the mold.

Principle 4: Casting hard-to-machine molds. Instead of machining the mold. cast a mold on a pattern. This approach is very interesting and convenient if the pattern is available (sometimes an original part that can be modified, a wax or plaster pattern that has been machined or a pattern made with some RP process). Many different materials to cast the mold can be used, depending on the number of parts that are needed and the dimensional tolerances required: Silicone rubbers, epoxies, polyurethanes (all these mixed with aluminum powder and/or other additives) and low melting point alloys such as Kirksite.

Principle 5: Avoiding the necessity of an ejection system. In SRP, the mold should be designed in such a way that the part can be extracted by hand, without ejection pins. The sprue, directly attached to the part, can be used to extract the part from the mold and then be removed. Special runners or additions to the part can be used in this same way. An ejection system significantly increases the cost of the mold, and increases the time needed to make it. Due to the fragility of some mold materials, ejector pins are definitively not used in these cases.

Principle 6: Avoiding the necessity of a cooling system. The machining of cooling lines can increase enormously the cost and the lead time to make the mold. Instead, the use of cooling plates (see Appendix B), forced air, and water mist sprayed on the external surface of the mold are recommended. If the material used for the mold has a low heat conductivity (epoxies, silicone rubbers, polyurethanes), it can be increased by adding high conductivity materials such as aluminum powder and aluminum chips. The thickness of these materials can also be reduced by casting them in a pocket in an aluminum block, which could be the mold itself, or an insert for the mold.

Mold design rules for SRP injection molding

Based on the principles for cutting costs in short-run production injection molding, the following rules were defined.

Rule 1: Apply each basic rule for designing molded parts, unless it is overriden by a rule for SRP injection molding.

Rule 2: Apply Principle 2 (Machine the mold in one operation) when designing a part.

Rule 3: Avoid to a maximum the use of ball end mills, tapered end mills and other special mills. Try to use only standard end mills (flat end mills). The use of ball end mills extends greatly the machining time of plane or quasi-plane surfaces. They can be used when the surface required is generated directly by the profile of the ball end mill.

Rule 4: There should be no external angles in the part. These external angles become internal angles in the mold and are not feasible with a simple machining operation. These angles should be changed to radii equal to available standard end mill radii.

Rule 5: The smallest external radius in the part should be equal to the smallest standard end mill radius to be used. In order to maximize the efficiency of the machining, this minimum radius should be as large as possible.

Rule 6: Eliminate the details in the part that can be obtained easily, quickly, and inexpensively in a secondary operation, especially those which are hard to make in a mold. Examples: Drilling a simple hole instead of having an insert in the mold, hot-bending a part instead of having a curved parting line, etc.

Rule 7: Cast the part details that can easily be machined on a pattern, but are hard to make in a mold and/or too lenghty to make as a secondary operation. Examples: Gear teeth, letters, logos, convex surfaces with standard ball end mill radii, etc.

Rule 8: Use standard elements as inserts. Avoid having to machine the inserts. Very often, standard elements such as pins, bolts, bar stock, etc., have already the desired shape and dimensions required and can be used as inserts with minimum modifications. Also, the design can be slightly modified so they can be used directly with a minimum of machining -or no machining- required.

Rule 9: Use standard bar stock, whenever it is possible, to make the core and cavity plates of the mold. Often, simple parts that do not require tight tolerances can be molded in such plates with almost no machining required for the faces that will form the parting line of the mold. Some manufacturers can also supply pre-ground plates and bar stock.

Rule 10: Include "ejection inserts" in the mold design. Whenever it is possible, use inserts as ejector devices. When there are not any inserts in the original design, include some "ejection inserts". These ejection inserts will facilitate the extraction of the part from the core plate, without damaging the part, or the mold.

Rule 11: Include "opening holes" in the mold design. These holes go through one plate, and match some blind holes on the other plate. The mold can be opened by inserting loose pins in these holes and knocking them gently with a hammer.

Some mold features design considerations

Because of their importance, several mold features are discussed in this section. In most cases, these considerations are valid for any kind of mold. Some of them are particular to short-run production molds.

The mold material. The injection molding process demands that the mold be sufficiently rigid and resistant to withstand the injection pressure and the clamping pressure. The clamping pressure basically is a function of the injection pressure and the projected area of the part to be molded. The temperature at which the melted plastic is injected will also affect the viscosity of the melted plastic and therefore will also affect the clamping pressure necessary to hold the plastic during the process.

The sprue. The size of the sprue at the nozzle end must be larger than the size of the nozzle orifice (about 1/32 of an inch). If several mold plates are used between the nozzle and the gate, this rule is valid for each plate (see Figure 3.1). In this dissertation, a mold base and a nozzle adapter are used and they will be considered as mold plates. The sprue hole must be smooth, free of scratches. It has been proven that a straight flute hand reamer gives the best results [28].

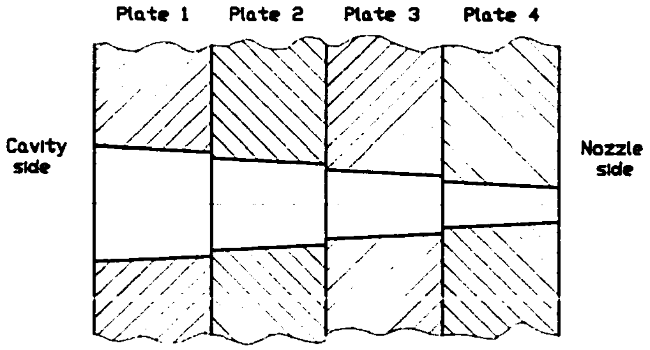


Figure 3.1 Aspect of the sprue when the mold is made of several plates.

The runners. In SRP injection molding. mold designers should try to avoid runners in order to reduce the machining time and the cost associated with it. If they are required. they should be as short as possible and sharp corners should be avoided. Although it would be ideal to just machine the runners on one of the plates to reduce the lead time to make the mold. the best runners have a round cross section (half round on each plate). Round cross sections reduce the pressure drop during the injection. allowing for relatively low injection pressures that are required for SRP molds.

Although runners are mostly used to distribute the melt in multi-cavity molds. they are also used to obtain a certain flow pattern inside the cavity. Almost all injection molded plastic parts will have some degree of molecular orientation. As the molten plastic enters and fills the cavity. it solidifies first on the relatively cool surface of the mold inducing flow lines patterns: then it will change slightly by the relaxation process. As described by Malloy [29]:

Residual Orientation = Orientation Level Due to Flow - Relaxation

When two flows of molten plastic in the cavity meet. they form a weld line. Weld line patterns and frozen flow patterns are also described by Malloy [29] and although the exact pattern and resulting characteristics are hard to define. it is relatively easy to predict where they will be and how they will affect the performance of the molded part. Because of that. it will be necessary sometimes to have runners and special gates in SRP molds.

The gating system. The objective of a gate is to let the melted plastic enter and fill the cavity as evenly and quickly as possible. The gate location, type and size have a great influence on the quality of the molded part and the pressure required to inject the plastic in the cavity. The choice of the type of gate is also a function of the viscosity and flow characteristics of the thermoplastic material. They are usually machined at last and undersized. If necessary, the gates can be enlarged and modified after the trial runs.

By experience, some rules of gating are:

- For symmetric parts the gate should be as near the center as possible.
- For parts that resemble a cup, a sprue gate or a reverse sprue gate is recommended.
- Avoid gates where the section is very thin.
- Always try to gate at the thickest section of the part. If necessary, add a heavy section to the part that can either be machined later, or left if it does not affect the aesthetics and function of the part.

Different types of gates are shown in Figure 3.2.

The vents. The vent system allows the air trapped in the cavity to escape when the melted plastic is injected.

Some authors like Schröer [30] think that venting slots are not really required in injection molds because the air can escape through mold plates and ejectors. He only mentions their necessity when side ejection is used or when the molded part is retained on the core by suction.

On the other hand, other authors think that under-venting is a common problem and that over-venting is rarely a problem and its most common consequence is that the part will have some flash, that can be trimmed [28]. Which can be acceptable for SRP, but definitely not for very large production runs.

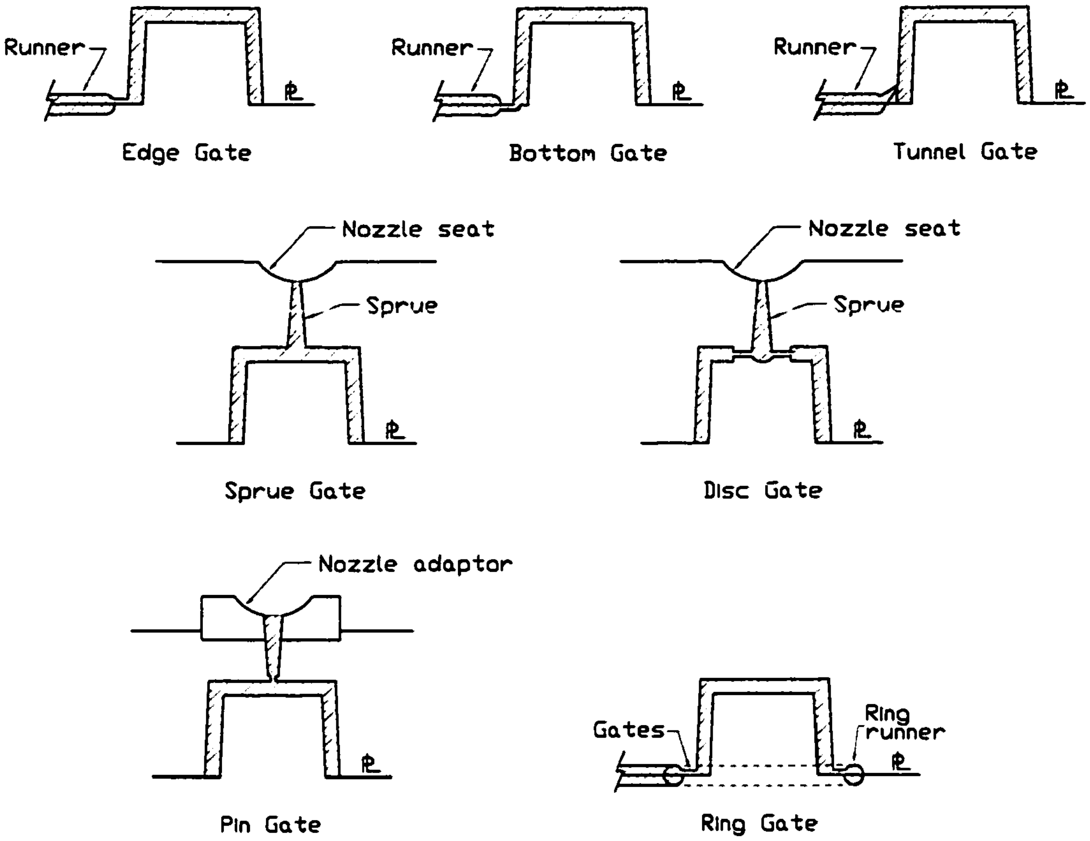


Figure 3.2 Typical sprues, runners and gates.

The fact is that a lack of proper venting can cause excessive pressure in the cavity and the air trapped inside will become extremely hot, leading to short shots, burn spots and other problems such as poor weld lines, awkward marks and high internal stresses [31].

Hartmann [32] also mentions that improper venting can cause bubbles of air in thin sections surrounded by thicker ones and in sections of the mold that are far from the parting line and far from vented ejectors. In such cases, appropriate vents must be included in the design and he presents several solutions.

Venting can be done through the parting line, core inserts, ejector pins, plates, and porous metal pins [33]. The choice of the proper combination of vents will basically depend on the geometry of the part and how the mold has been made.

The most common vents are those that are machined at the parting line. They have to be machined at the opposite side of the gate: The larger the gate, the larger the vents. The width and number of vents is defined by the size and geometry of the cavity and the speed at which it is filled with the plastic. Vents actually consist of a primary vent and a secondary vent, also called vent relief (see Figure 3.3).

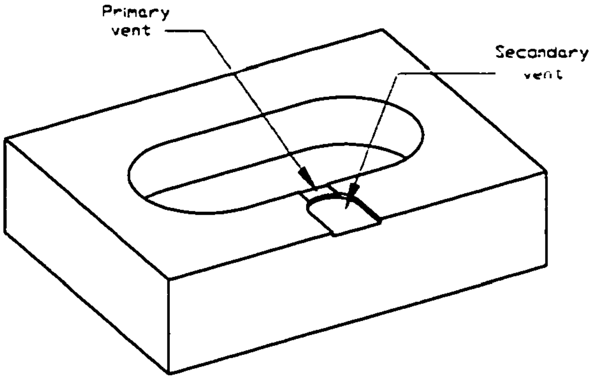


Figure 3.3 The vent consists of the primary vent and the secondary vent.

The primary vent starts at the cavity and has a length of 0.03-0.25 and a width that can vary a lot; the secondary vent goes from the primary vent to the edge of the mold plate and its width ranges from 0.25 to 0.50 inches. The depth of the primary vent depends a lot on the resin being injected, its temperature and pressure. In Table 3.2, there are some recommendations made by the Society of Plastics Engineers (SPE) [34]. The depth of the secondary vent is typically 0.03 inches.

If the part is relatively deep, or it has many ribs it will probably be necessary to have vents at other places too: Core pins and core inserts can be used to that effect.

Mold opening. Frequently, class "C" molds will be hard to open. The part will frequently stick to both plates and keep the mold closed. This will especially be true when the sprue in the cavity plate of the SRP mold is not tapered. It will also happen when there are some inserts on both mold halves.

To open the mold easily, without damaging the parting line surface, four holes can be drilled. These holes should go through on the mold half with the dovetail slot, and be blind

Table 3.2 Vent depths for various resins (inches).

RESIN	Minimum	Maximum
ABS	0.0010	0.0015
ACETAL	0.0005	0.0010
ACRYLIC	0.0015	0.0020
CELLULOSE ACE., CAB	0.0010	0.0015
ETHYLENE VINYL ACET.	0.0010	0.0015
IONOMER	0.0005	0.0010
NYLON	0.0003	0.0005
PPO/PS (NORYL)	0.0010	0.0020
POLYCARBONATE	0.0015	0.0025
PET/PBT/POLYESTERS	0.0005	0.0007
POLYSULFONE	0.0010	0.0020
POLYETHYLENE	0.0005	0.0012
POLYPROPYLENE	0.0005	0.0012
POLYSTYRENE	0.0007	0.0010
POLYSTYRENE(Impact)	0.0008	0.0012
PVC (Rigid)	0.0006	0.0010
PVC (Flexible)	0.0005	0.0007
POLYURETHANE	0.0004	0.0008
SAN	0.0010	0.0015
T/P ELASTOMER	0.0005	0.0007

on the other mold half. By placing loose pins in them, and knocking them gently, the mold can be safely opened.

Ejection of parts. As it will be shown later, in chapter 5, some mold inserts can be used as "ejector pins". Ejection of parts in SRP molds is delicate because of the relatively soft materials used to make the molds. The ejection will also be harder, since tapered end mills are avoided in the machining process. To overcome this problem, two simple systems are proposed: The "ejection inserts" and "pullers".

An "ejection insert" is basically a standard ejector pin, but used in reverse. The head of the ejector pin is located in a counterbored hole in the mold. Custom made ejection inserts can be made when considered more appropiate. Brass, or aluminum, can be used in this case.

A "puller" is basically an extra detail in the mold that sticks out when the mold is open. By pulling it, the rest of the part is extracted. The "puller" is removed after the extraction of the part.

Heat control. The mold has to be able to extract the heat of the melted plastic at a reasonable rate in order to obtain a part with the mechanical properties and surface finish required. Defects can occur either because there is too much or too little heat extraction. Ideally the mold should be very close to the melting temperature of the plastic during the injection and when the mold is filled, it should cool down as evenly and quickly as possible to freeze the plastic. This is a very delicate parameter that is hard to calculate exactly and the heat extraction rate is usually refined during the trial runs and must actually be redefined constantly during the process until a steady state heat transfer is reached. And it takes several hours and a very stable and constant production to get to that point.

The heat transfer of the molds will depend greatly on the conductivity of the tool material and the cooling/heating system. In general, cooling lines are not common for SRP tools, because the cost of making them exceeds the cost of having longer cycles. Also, when brittle materials are used for the mold, it reduces the resistance of the mold, as it is the case in Direct ACES Injection Molding (DAIM).

Flexible and modular fixturing for SRP mold machining

Background on current fixturing technology

A fixture is a device that performs the work-holding duties in a manufacturing fabrication or assembly operation. A flexible fixture is one that is readily programmed or adapted for a variety of parts or products, as opposed to a special-purpose or custom fixture. Turning fixture design into a process means defining general fixturing design rules and methods, that can be included in a CAD package [35].

In the past fifteen years, the average setup cost has dropped from 20% to 13% of the total production cost. This has been possible thanks to modular and flexible fixturing systems. Yet, although close to half of all job shops use some type of modular fixturing, they use it only 10% of the time [36].

Research in this field has been very active, and recent efforts have been focused toward developing alternative solutions to traditional fixturing technology. The optimal selection of support positions, the configuration of the fixture and the automated generation of the fixture have been the subject of many research studies. Some systems have been proposed by Menassa and DeVries [37] and by Rong and Bai [38] for fixturing prismatic components. In the case of the SRP molds discussed in this dissertation, the position of the counterbored holes for fixturing the mold can be defined by a simple top view projection of the mold. This is actually a particular case of the automated method described by Trappey and Matrubhutam [39].

In Figure 3.4 a schematic diagram shows the current approaches being investigated and developed [40].

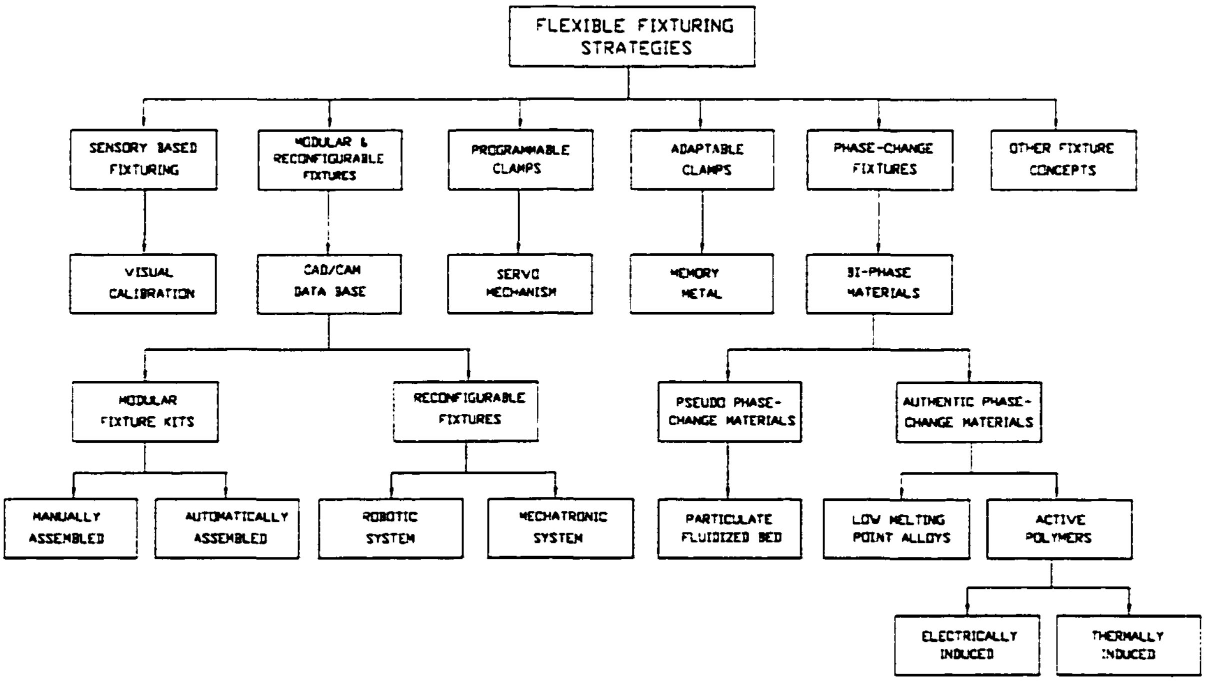


Figure 3.4 Flexible Fixturing Technologies being used or investigated.

Fixture for machining a short-run production mold

The fixturing system to machine an SRP mold must allow the CNC machine to do several operations without moving the workpiece. These operations are:

- Machine the surface that will define the parting line of the mold.
- Drill and ream the holes for the locating pins.
- Machine the mold cavity.
- Machine other mold details such as ejector pin holes, sprue, runners, gates.

To accomplish this, several fixturing technologies are available. Two interesting options are described:

Fixture-free machining. An interesting approach to manufacture the mold in one operation, is to use the Fixture-free machining technology described by Hanada, Bandyopadhyay and Hoshi [41]. In this technology, block-like materials are machined from round stock mounted in an indexing head housing. By rotating the head 90, 180 and 270 degrees, all circumference faces can be machined easily (face milling, drilling, pocketing, contouring, etc.), without disassembling the workpiece. The remaining faces are generated when the part is cut off with an end mill. This process can be completely automated, and CNC code could be generated by special post-processors, from the CAD model. Probably the only inconvenience of this approach is that a large amount of machining is required to obtain the six faces of a mold half.

Proposed fixturing system for SRP mold machining. The simplest fixturing approach is to have an elevated plate with a grid of holes mounted on the CNC machining center table. The block to be machined will be mounted on that plate with socket head screws. The previous preparation of the block is very simple: Assuming that the bottom of the block is sufficiently flat, only two counterbored holes need to be drilled at some predetermined positions so they are aligned with the holes on the plate and they are outside the cavity (or core) to be machined. These holes do not need to have a tight tolerance since all datum surfaces and features will be machined in one operation. The tolerances on all dimensions will basically defined by the accuracy of the CNC milling center used. Although this fixturing system requires that counterbored holes be machined on the workpiece, in the context of mold-making it could be said that it is very close to be a flexible fixturing system. In any case, it is simple and gets the job done quickly. Figure 3.5 shows a prototype fixture that was built for this purpose. This particular fixture can be mounted in the vise of the VM40 Seiki CNC Milling center at the Engel Lab.

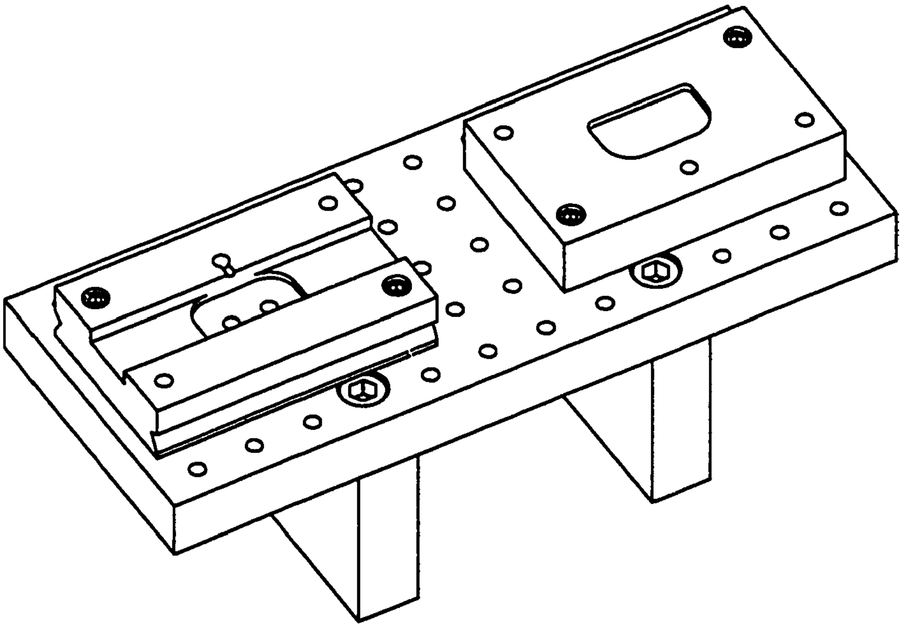


Figure 3.5 The flexible fixturing system proposed for SRP mold machining.

4 MOLD BASE DESIGN FOR SHORT-RUN PRODUCTION

Short-run production injection molding requires an injection molder as flexible as possible. The most flexible systems available now are vertical manual injection molding machines. These machines, although relatively small, frequently desktop systems (see Figure 4.1), are the best solution for companies or service bureaus dedicated exclusively to rapid prototyping or production runs of up to about 500 parts.

In these machines there are no mold bases. The molds usually consist of just a core plate and a mating cavity plate. The mold is previously assembled on the workbench, located on the press table, clamped and the injector nozzle is directly in contact with the mold when the plastic is injected. This is the ideal solution for this type of production because of the following reasons:

- The fixed costs are relatively low. The cost of this type of machine is lower than large production automatic machines; these systems are simpler to maintain; downtime of these machines is less expensive.
- There are no special costly peripherals. Although basic, these machines can provide good monitoring and control of the injection molding process.
- They are extremely flexible. The fact that they are operated manually makes them flexible *per se*; the whole process is operated, monitored and controlled by the most flexible component of any system, the human being. In this particular case, it is definitely an asset.
- Actual throughput may be higher than in automatic machines. For Short-Run Production (SRP) and Rapid Prototyping (RP), a manual machine is consistent with today's

"Theory of Constraints" and "Just-In-Time" policies in many industries. For example, several different parts can be molded in the same machine, provided that they require the same injection material. Cycle times being sometimes in the order of minutes for some RP molds, they can be clamped separately so that they can be removed from the machine after the plastic is solidified at the gate. While they cool down on the workbench, another mold can be placed in the molder and so on. In an automatic machine, this could not happen.

- Although very high injection pressures are not attainable, this is usually not a problem in SRP and RP injection molding, because the molds themselves cannot withstand high pressures.

Many laboratories and R&D departments of large companies and service bureaus have similar machines for their prototyping needs, but the majority of plastics injection molding companies, mainly focused on large production runs do not have them. The challenge in this case is to transform an industrial horizontal injection molding machine (see Figure 4.2), into a more flexible machine that can also handle short-run productions at an affordable cost. In the following section, some solutions to make this process more flexible, in particular the use of standard frames and mold bases, are presented.

Characteristics and function of standard mold bases

Nowadays, most molds are made using as many standard elements as possible to simplify the design process and reduce their cost. This includes, among others, sprue bushings, ejector pins and plates, hot-runners, heater elements, master frames and standard cavity plates.

A mold base (Figure 4.3) is actually a set of standard mold components that will allow the mold designer to focus his attention on the cavity plates and the cavity and core inserts they will hold. The other components such as the "U" frame, the ejection set and support plates are already defined and standardized.

The front half of the mold base (Figure 4.4) is located on the stationary platen of the injection molding machine. It consists basically of the clampling plate with the sprue bushing

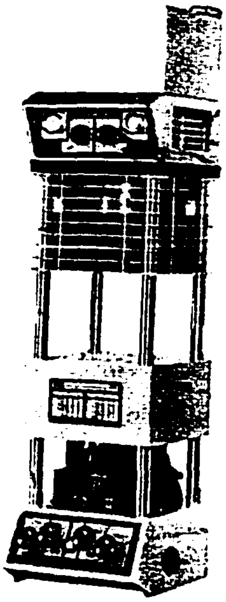


Figure 4.1 Desktop injection molder.

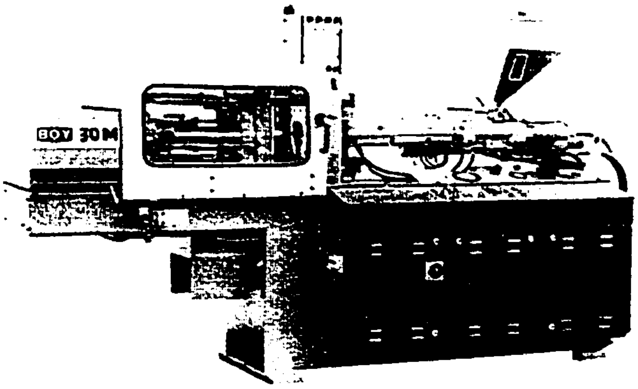


Figure 4.2 Plastics injection molding machine in the Engel Laboratory. at Iowa State University.

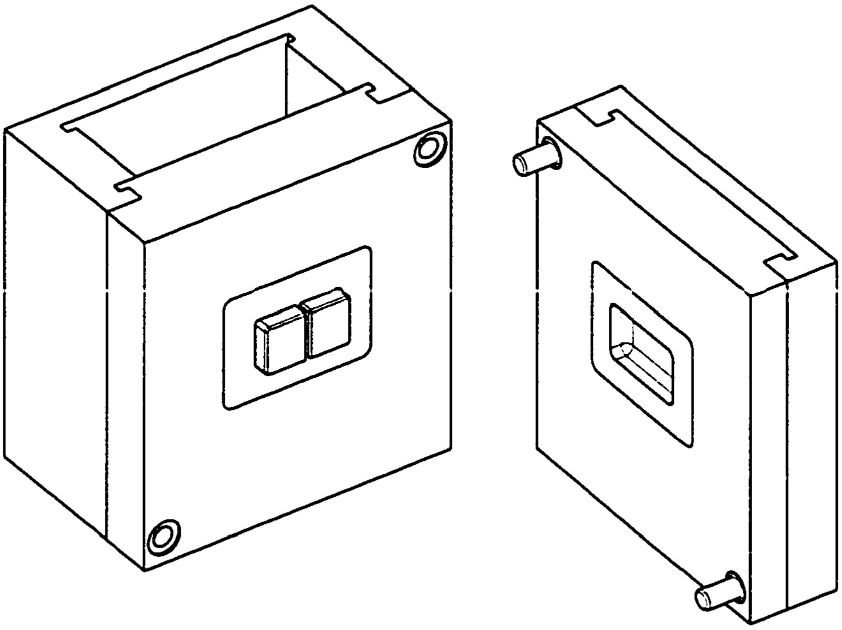


Figure 4.3 The rear and front halves of a mold base.

and the front cavity plate (plate A), that holds the cavity inserts and the leader pins that will maintain the alignment of the mold halves.

The rear half of the mold base (Figure 4.5) is located on the movable platen of the injection molding machine. It consists basically of the "U" frame and the rear cavity plate (plate B) that holds the core inserts and the leader pin bushings.

The U-frame consists of the rear clamping plate, the spacer blocks and, sometimes, support pillars. This U-shaped structure provides the space for the ejection system assembly.

The ejection system consists of two ejector plates that hold the knock-out pins and return pins. The return pins actually guide and support the ejection system. When the mold base is closed, plates A and B will mate and they will therefore form the parting line of the mold cavity.

The cavity plates can be as simple or as complex as they need to be. For very simple parts, with a flat surface, only plate B will be machined, directly. Plate A will only hold the sprue. For very complex molds, they will have cooling lines, ejection sets, several cavity and

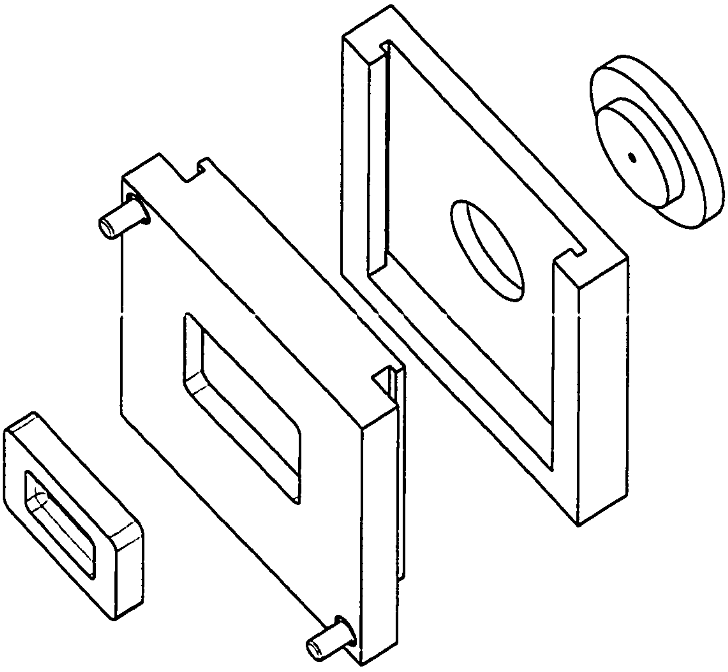


Figure 4.4 Components of the front half: Cavity insert, plate A, clamping plate and sprue bushing.

core inserts, slides, hot runner systems, temperature sensors, limit-switches and more. But the main function of the cavity plates is to contain and align the core and cavity inserts.

In general, the cavity plates are used only once, for a specific mold. In some cases, they can be used for several different parts, provided that there has been some kind of "in-house" standardization of the inserts held by these cavities. Also, if a mold has become obsolete, they can be reused assuming that the previous machining does not interfere with the new configuration of the mold. Lately, some mold and die manufacturers have come to new ideas to make mold bases more flexible and to lower the costs of making them.

The first approach was to design a quick-change system for the whole mold base. Such a system, patented by Martin [42] and assigned to Master Unit Die, Inc. (MUD), consist of clamp plates on the platens of the press with slots where the mold base can be engaged.

The next step was to extend this idea to the U-frame: In such a system, the U-frame is mounted on the movable platen of the press and the cavity plates are interchangeable. This

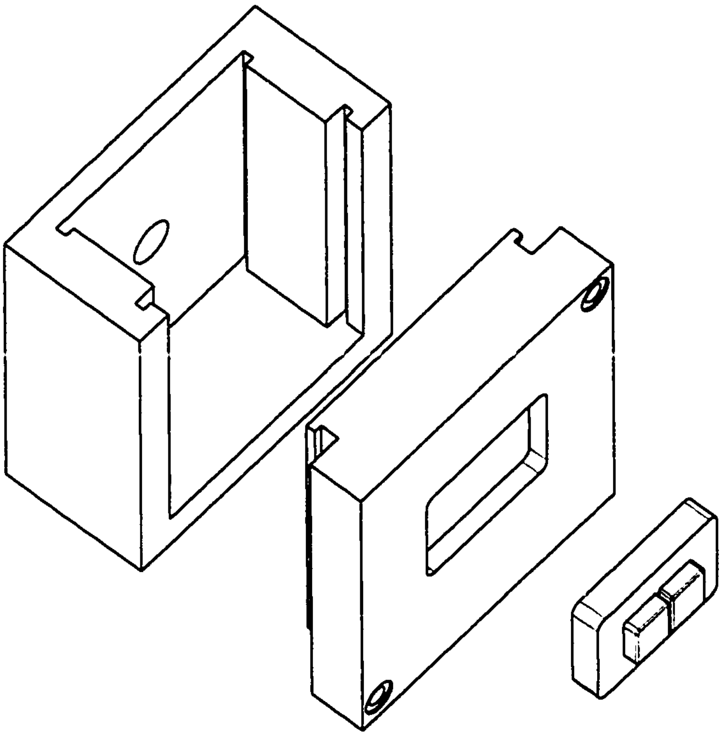


Figure 4.5 Components of the rear half: U-frame (clamping plate and spacer blocks), plate B and core insert.

system, also used by MUD, is illustrated in previous Figures 4.3, 4.4 and 4.5. In this case, the cavity plates slide vertically in slots located on the clamping plate and the U-frame. An ejector system and cooling system can be used with this design. Although a great improvement to make the process more flexible, the plates are still quite heavy and it is relatively expensive for SRP. It is an excellent solution for medium to large-production runs though.

Another improvement, reducing even more the required machining of the cavities and the weight, is to have a standard mold base with interchangeable inserts. Such an approach is used by Pleasant Precision, Inc. [43] in a very interesting patented design, which is called *Round Mate* [1]. In their mold system (see Figure 4.6), the inserts are round, self-aligning and they have an integral cooling and ejection system. There is no need to connect anything else: In a matter of minutes they can be replaced, on the press, by one person.

Note that in this case the steel inserts have a standard sizes (diameter and depths) and

---

[1] Round Mate is a Trademark of Pleasant Precision, Inc.
U.S. Patents Nos. 4,828,479 4,959,002 5,261,806 5,647,114 and other patents pending

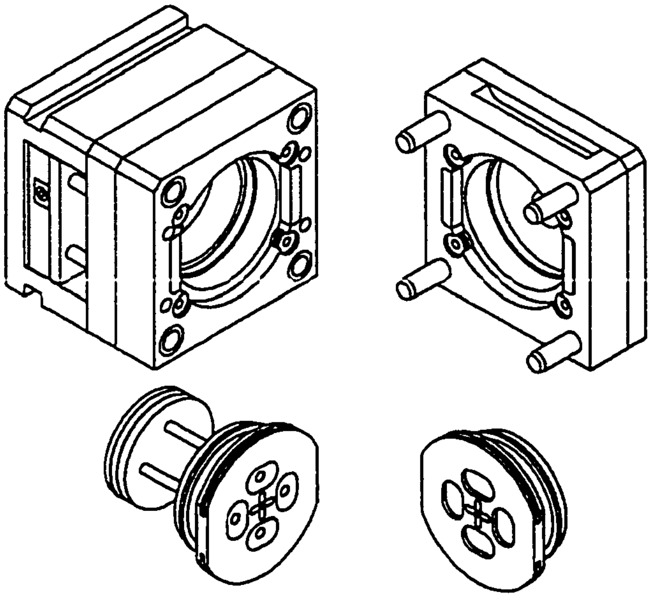


Figure 4.6 *Round Mate* Interchangeable Insert Mold System with integral cooling and ejection set. (*Courtesy of Pleasant Precision, Inc.*)

can be used either for SRP or long-run production.

Another approach is known as "Modular Molding" [44] in which small dissimilar parts, frequently from different costumers, can be injected in the same mold base for SRP. In this case, plates A and B have slots for the cavity inserts and a permanent sprue and runner system. The inserts can be placed and locked into position with special key/lock devices without removing the mold frame from the press. Each insert can have several gates coinciding with some runners on the cavity plates and is designed in such a way that it will close the unused runners. The resulting ejected part consists mostly of the frozen plastic in the sprue and runners with some of the runners ending in the desired molded parts. The plastic sprue and runners can be shredded and recycled. Although not very efficient in the sense that there is a lot of scrap, it is adequate for SRP and prototyping. Note that in this case, the inserts have standard width and depth and can vary in length.

As it can be observed, in order to make a mold base more flexible, the tendency is to reduce the manufacturing process to the core and cavity inserts, just as it is done in the simplest molds used in manual presses. In the following section specific characteristics required for a flexible mold base are presented.

Characteristics and function of short-run production mold bases

The main function of a short-run production mold base is still to hold and align the cavity and core halves of a mold. It also must have the following characteristics in order to be a low-cost, flexible tool:

- It has to be able to hold different sizes of cavity and core inserts, without the necessity to machine it.
- It must be able to align the cavity and core inserts within a specified tolerance.
- It must be able to locate and hold in position the cavity and core halves.
- It must allow the process to be completely manual for very short-run production, allowing the operator to easily place and extract the mold from the press in order to assemble or disassemble the mold on the workbench.
- It must allow the process to be semi-automatic, for medium-run production, allowing the possibility of using a cooling system and an ejection system.

To comply with these characteristics, a SRP mold base was designed, starting from a standard mold base from Master Unit Die, Inc. The mold base consists of the Quick Change "U" style frame model 84/90 UF, already clamped on the platens of the Boy 30M injection molding press at the Engel Lab., and the solid construction "T" style interchangeable insert molds model 84/90 TSU. The interchangeable insert mold, as it is called by the manufacturer, is actually both the support plate and cavity plate in one element. In this thesis, the front and rear interchangeable mold inserts will be referred to as "plate A" and "plate B" respectively (see Figure 4.7).

Plate A comes in different thicknesses ranging from 1.500 to 2.625 inches, and plate B from 2.000 to 2.625 inches. In this particular design, plate A and plate B have thicknesses of

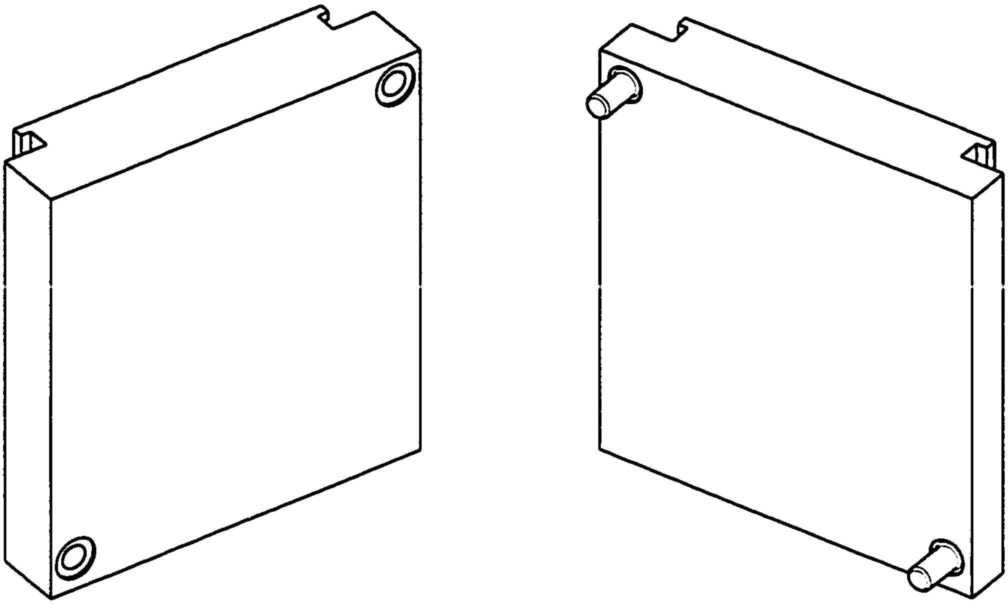


Figure 4.7 Plate B and plate A, before machining.

2.000 and 2.250 inches respectively. These thicknesses do not have a specific purpose but it is obvious that a thicker plate will allow molded parts with larger depths. In any case, the design concept presented here does not depend on this parameter.

The characteristics for a short-run production mold base described previously lead to the following design requirements and solutions:

- In order to allow different sizes of molds, the pockets or slots in the plates must be as wide and deep as possible. In this case, horizontal and vertical slots, having the same depth, were machined on the plates. With this, inserts longer than the plates themselves can be placed either vertically or horizontally.
- In order to maintain the alignment of plate A with respect to plate B, the original leader pins and bushings were kept. In this case MUD's plates have only two leader pins diagonally placed. In order to keep the symmetry of loads when the mold base is clamped without the inserts, the other two corners were also kept. In a way, by machining the

plates this way. the design was equivalent to having a support plate with pillars (the four corners) and locating pins. These pillars. actually will not support any load when the mold is mounted on the plates and clamped for injection molding. In this particular case. the combined thickness of the mold halves must be larger than 1.95 inches. which is equal to the sum of the depths of the slots in plates A and B. and smaller than 2.50 inches. to allow a minimum leader pin penetration depth of 0.50 inches.

With this kind of setup. two different alignment are required:

1) the alignment of the sprue in plate A with the sprue in the cavity insert. and

2) the alignment of the core and the cavity inserts.

The latter is definitely more important since a misalignment of the core and cavity inserts usually leads to a defective molded product. To assure this alignment. the core and cavity halves will have to have alignment pins. or some other mean to accomplish the alignment. as it was discussed in chapter 3. The alignment of the sprue will allow some error since it will not affect the product itself and this fact is very important because it will add robustness to the design.

A fixturing system to locate and hold the mold during the injection process is needed. and it will have the following objectives:

- It has to be flexible in the sense that it must allow for accurately locating the mold almost anywhere within the plate:
- it has to be rigid enough to hold the mold in position during the opening and closing of the press:
- it must allow the operator to easily place and remove the mold from the plate during manual operation:
- it must assure the repeatability of the location of the mold during manual operation: and

- it must rigidly hold in place the core half of the mold during semi-automatic operation.

The fundamentals tasks of fixturing [45]: Locating, supporting, clamping and referencing the tool to the workpiece (in this case, aligning the sprue holes in plate A and the mold) are also applied. Based on the objectives defined for the fixturing system, a flexible fixturing approach was adopted since these objectives are basically those of a flexible fixture, as defined by Boyes [46].

Flexible fixturing, and more specifically modular fixturing, has been applied almost exclusively for manufacturing processes such as machining, gaging and welding but not for injection molding. The 3.2.1 principle for positively locating a workpiece [47], in this case the mold, is still valid. But in this new field for modular fixturing, some differences exist and will define new constraints: First, the forces applied to the workpiece (in this case the mold) are in only one axis, they are known, and are relatively very high; second, no torques are applied to the mold. In such conditions, it is clear that only a solid and rigid planar base can support the loads: In this case, this base is the plate B. The other requirements for the fixture are to locate the mold on plate B and hold it there when the mold is open. Since the press is horizontal, it must also prevent the mold from falling. This would not be a constraint in a vertical molding press, where the mold can just be positioned by a 3.2.1 locating device and the gravitational forces will hold it in position until the press clamps the mold. Also, as another restriction, the fixturing system must require the minimum amount of machining on the mold; ideally, none.

An interesting approach for this case is the use of particulate bed fixtures to hold the mold halves in position. Flexible particulate bed fixtures have been extensively studied by Abou-Hanna, Okamura and McGreevy [48, 49, 39], and this fixturing system was considered as a possible way to mount the mold halves on the platens of the molding press. Two facts about this fixturing system made it very attractive as an option to hold in place the mold halves: First, it can hold about any shape; and second, the molding press can provide the compression required by this fixturing system. After considering problems related to the alignment of the sprue and problems related to the parallelism of the mold plates and platens of the press this approach was discarded as impractical for the purpose of this research. It is recommended that further study be done in that field though since some interesting characteristics of this technology could be applied to very-short-run tooling.

The fixturing system chosen for SRP molds

To comply with all the described constraints, a solution in which the mold is engaged into an adjustable dovetail slot, was chosen.

The reasons for choosing a dovetail-type slot approach are that:

- It allows the mold to be guided but it prevents the mold from falling (only one degree of freedom).
- the machining requirements on the mold are simple and fast.
- a simple stopping feature is needed to positively locate the mold on the plate.
- it allows for different sizes of molds.
- the system is modular and simple.

In order to satisfy all the described constraints, a grid of holes was drilled on plate B and special "dovetail clamps" and "stop clamps" were designed. With this arrangement, molds of different sizes can be placed and held in position almost anywhere on the plate. Standard hexagonal bolts, washers, and square nuts are used. The length and width of the slots on the dovetail clamps assure a robust, adjustable, and accurate positioning of the mold. Slots in the back of the plate allow the operator to use one hand to slide and hold the clamp in position and the other to tighten the bolts. The grid of holes can also be used for a simple ejection system or to directly bolt very large mold inserts on the plate. A close-up view of plate B with the grid of holes and the dovetail fixturing system is illustrated in Figure 4.8. In Figure 4.9, some details of the dovetail clamp are emphasized. A partial section of the clamped mold mounted with the dovetail fixturing system is shown in Figure 4.10.

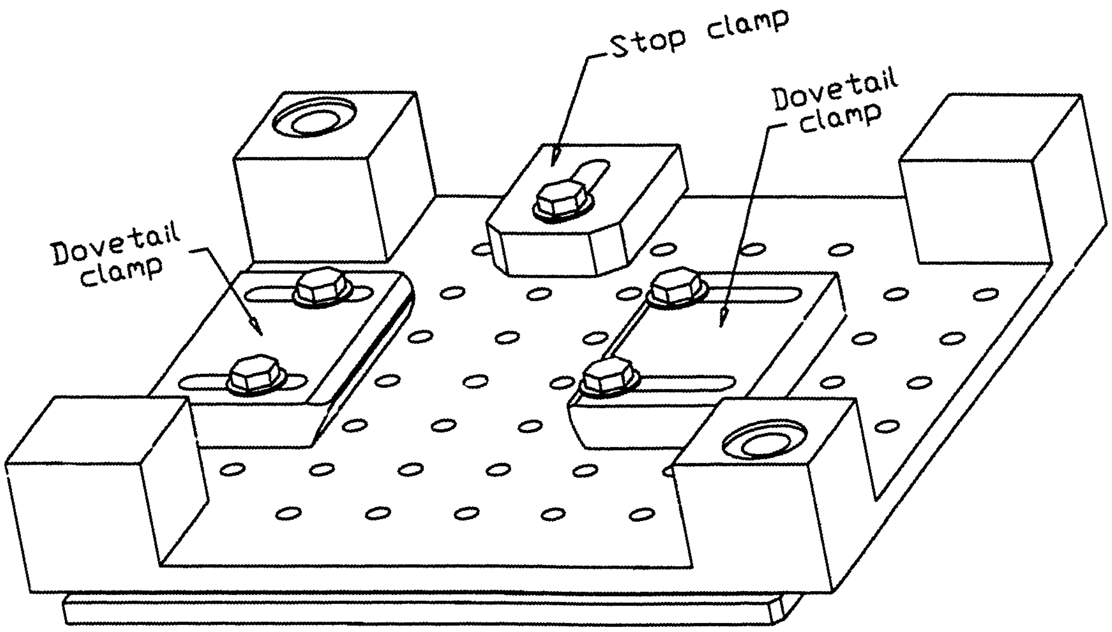


Figure 4.8 Close up view of the dovetail-type flexible fixturing system.

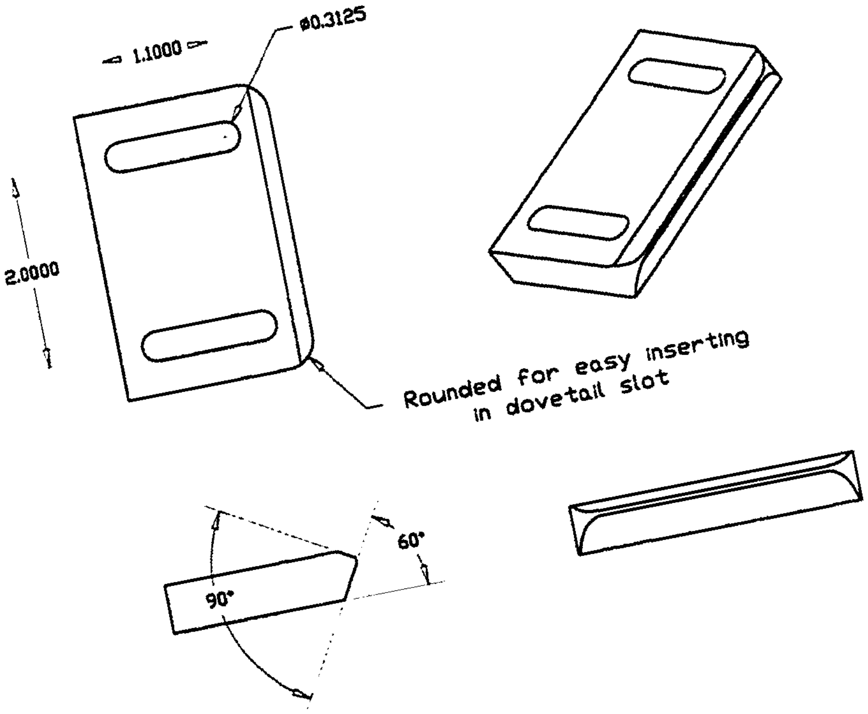


Figure 4.9 Details of the dovetail clamp.

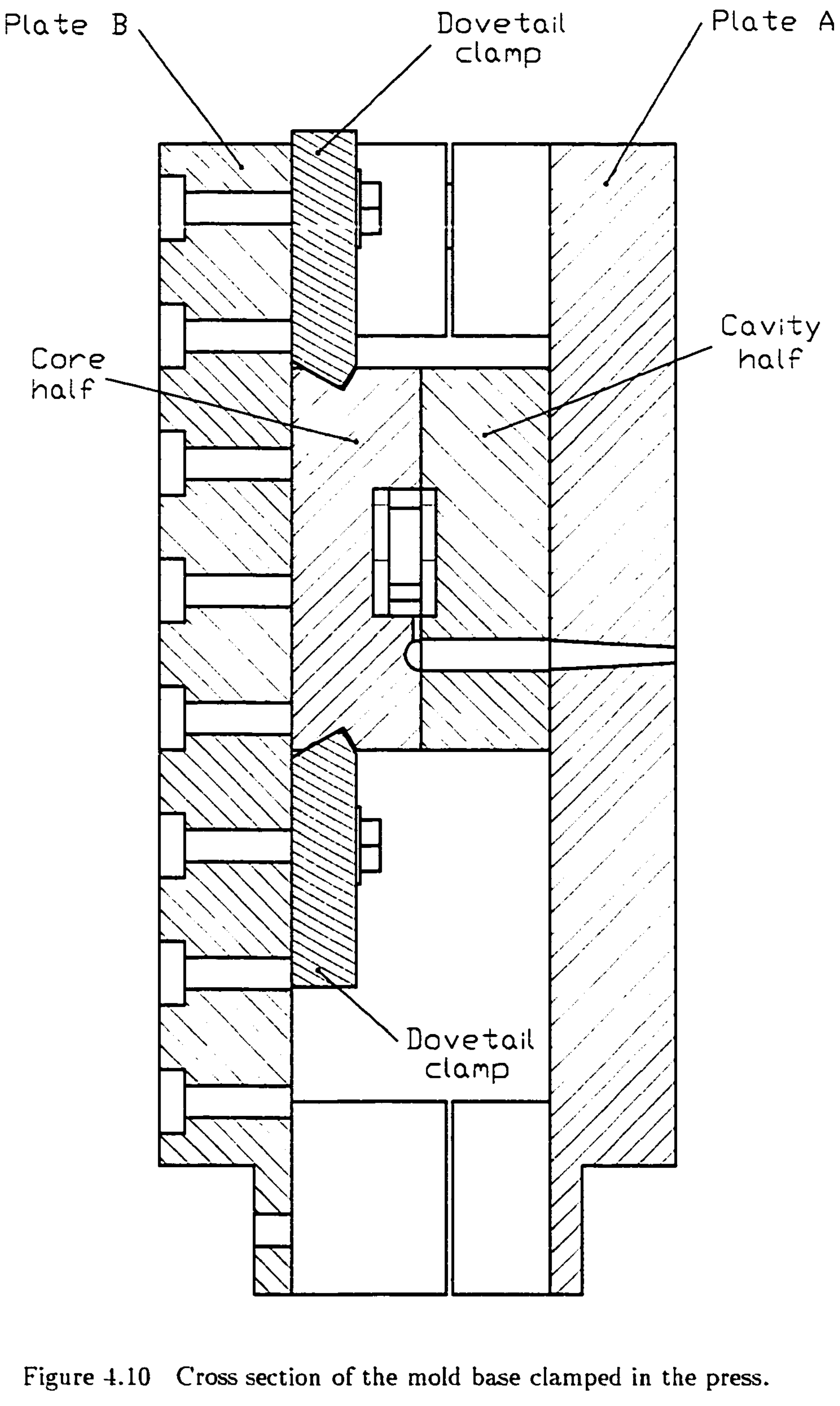


Figure 4.10 Cross section of the mold base clamped in the press.

For manual operation, the only machining required on plate A, besides the slots to accommodate the cavity half, is the sprue hole. For semi-automatic operation though, a grid of holes and slots in the back of plate A, similar to those on plate B, are also needed to locate and clamp the cavity half. Figure 4.11 and 4.12 show the front and back of plates A and B for semi-automatic operation.

In Figures 4.13 and 4.14 the mold base plates for a manual operation show how the mold halves are first assembled on the workbench and then it is slid into the dovetail slot formed by the two dovetail clamps. The clearance must be adjusted once so the mold can slide easily and still assure that there will be a good enough alignment of the sprue holes in the cavity plate and plate A.

Another way to run the process manually is to leave the mold cavity on the movable half of the mold base by completely clamping the core half on plate B. In this configuration, the mold is assembled and opened on the press, instead of on the workbench (see Figure 4.15). Ergonomically speaking, this maneuver is not comfortable, but this alternative might suit some specific cases where the molded part comes of easily.

For a semi-automatic operation, the mold cavity is located and clamped on the movable mold base and the clamps are loosely positioned in the fixed mold base as can be seen in Figure 4.16.

The mold core and mold cavity are then assembled with the small dowel pins. After assembling plates A and B, the cavity half is completely clamped to plate A. The mold base is then opened and the small dowel pins are removed from the mold halves. Figure 4.17 shows the assembly ready to be mounted in the injection molding machine for semi-automatic operation.

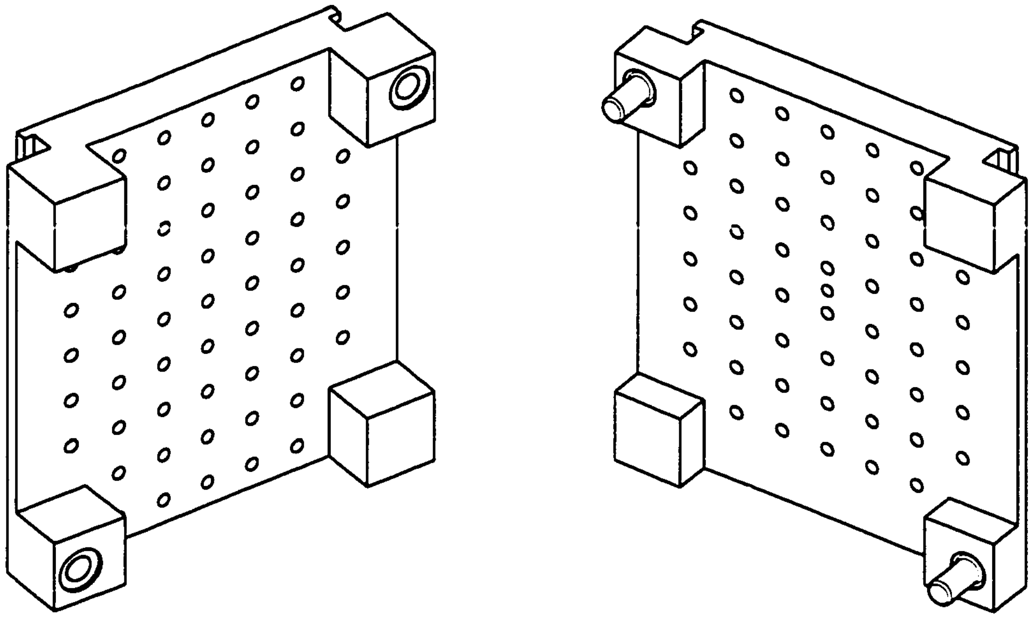


Figure 4.11 Front of plate B (left) and plate A (right), after machining.

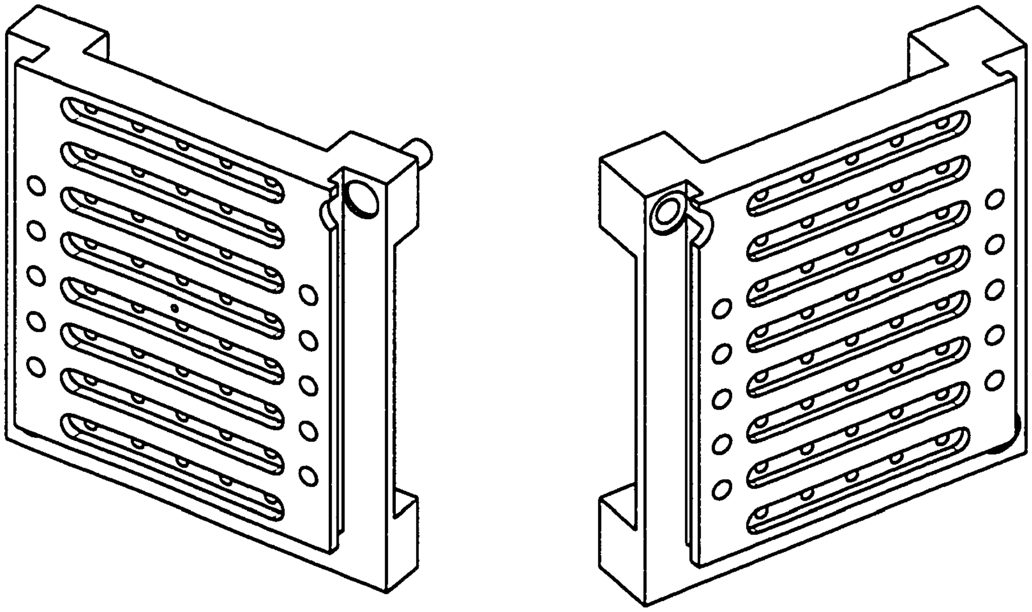


Figure 4.12 Back of the mold base components, after machining.

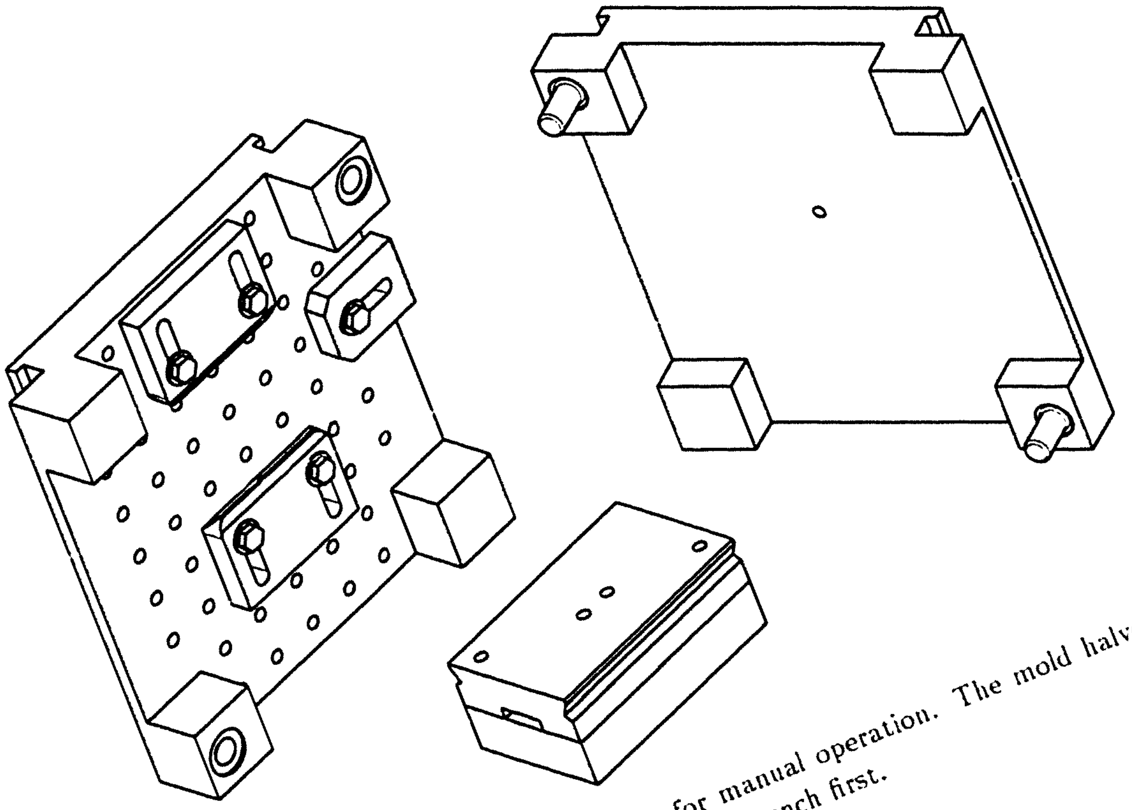


Figure 4.13 Mold base assembly for manual operation. The mold halves are assembled on the workbench first.

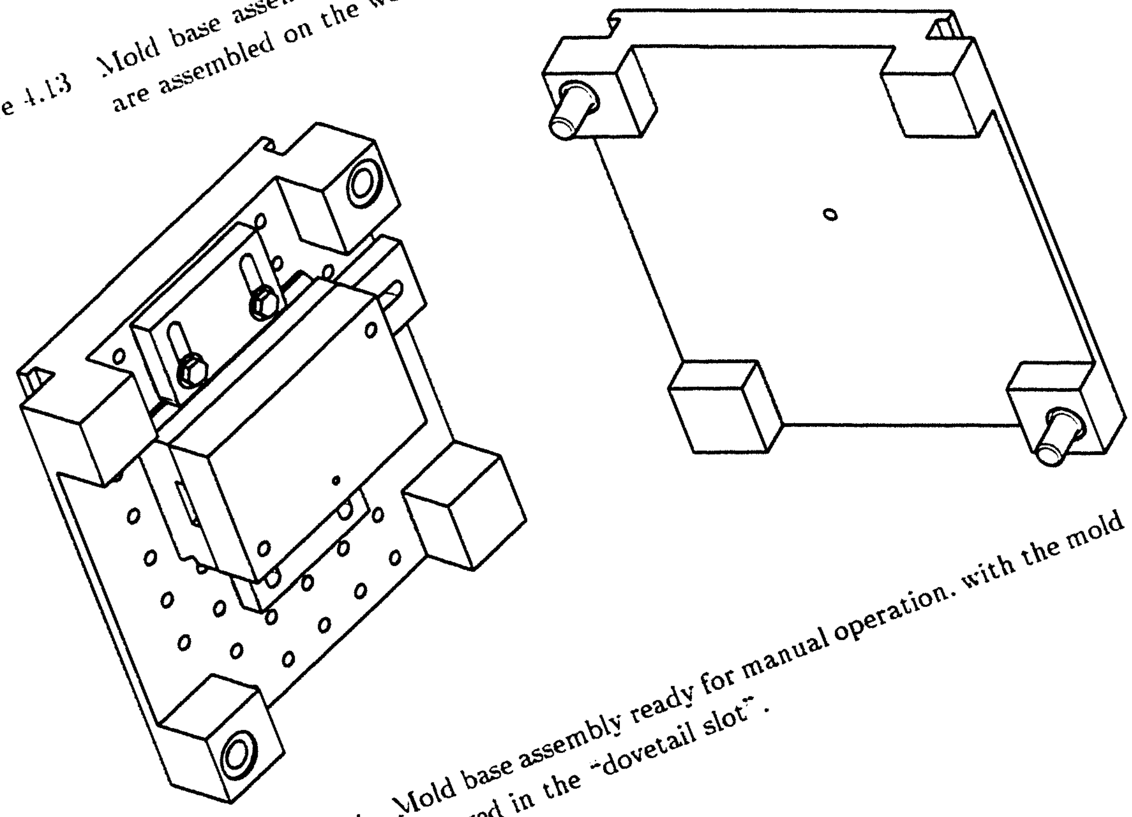


Figure 4.14 Mold base assembly ready for manual operation, with the mold engaged in the “dovetail slot”.

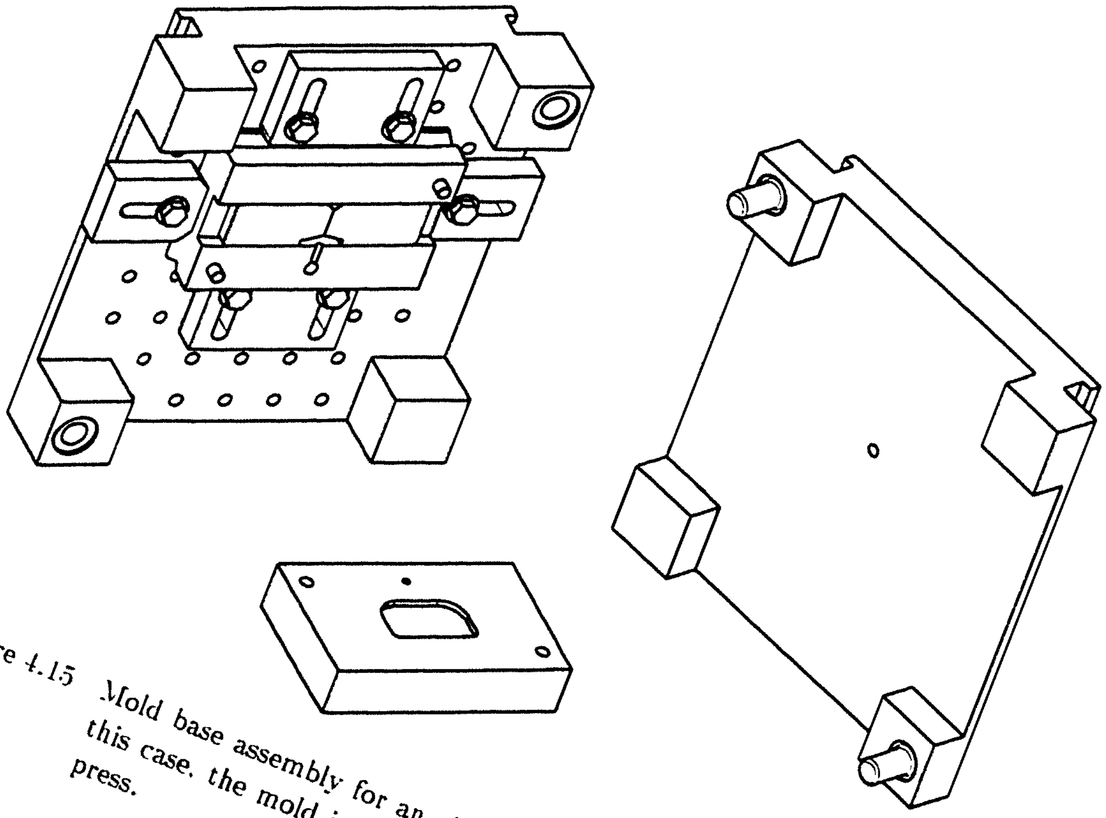


Figure 4.15 Mold base assembly for an alternative manual operation. In this case, the mold is assembled and opened on the molding press.

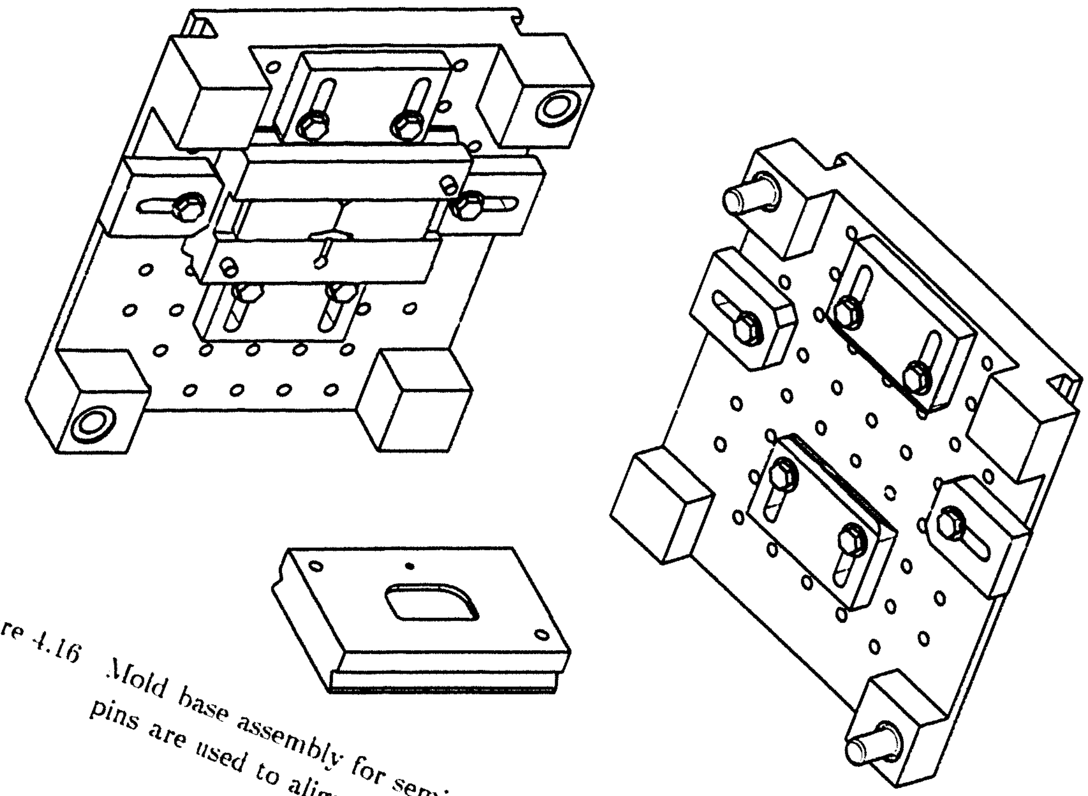


Figure 4.16 Mold base assembly for semi-automatic operation. The small pins are used to align the mold plates, and then removed.

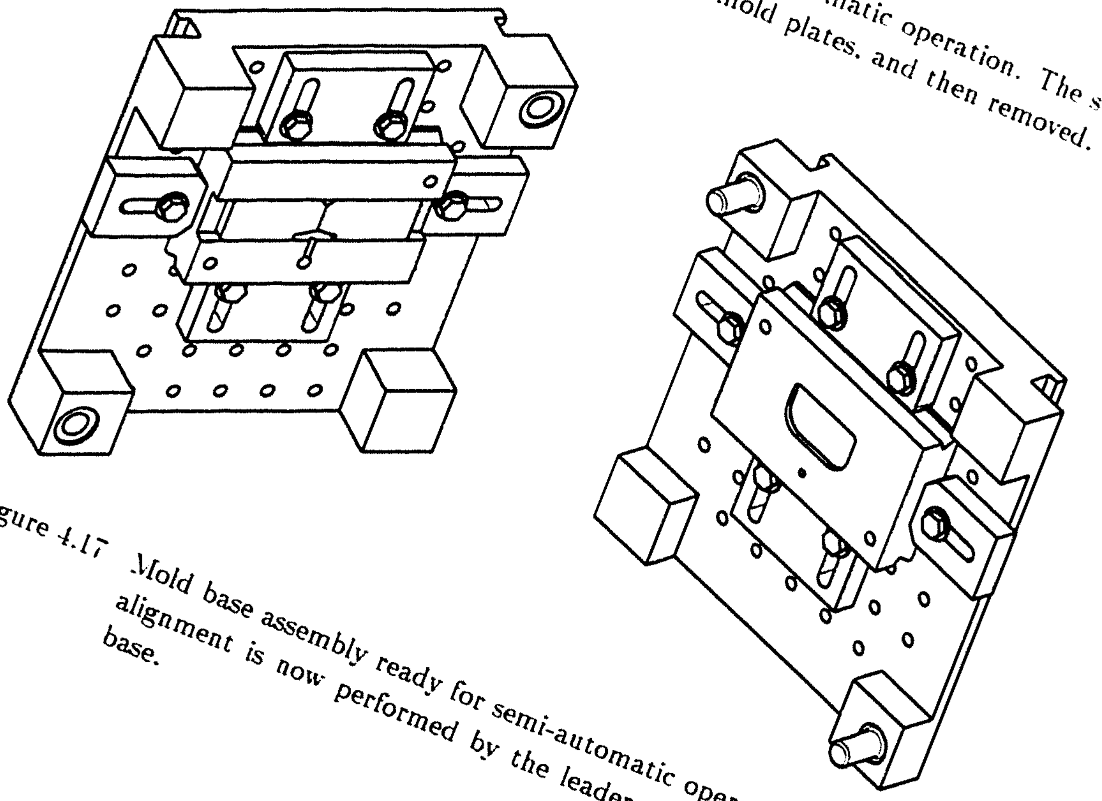


Figure 4.17 Mold base assembly ready for semi-automatic operation. The alignment is now performed by the leader pins of the mold base.

5 DESIGN FOR RAPID MOLDING

In this chapter, the complete design of a plastics injection mold will be done, from the redesign of the part, following the rules and guidelines defined in chapter 3, "Design Rules for Short-Run Production Injection Molds", to the evaluation of the actual injection molding process, the prototype parts, and the mold itself.

A study case: Knee brace central component

In this case, the part to be made is the central element of a knee brace used by foot-ball players at Iowa State University (see Figure 5.1). The material used in the original part was known to be a glass fiber reinforced Nylon. The actual content of glass fiber is not known.

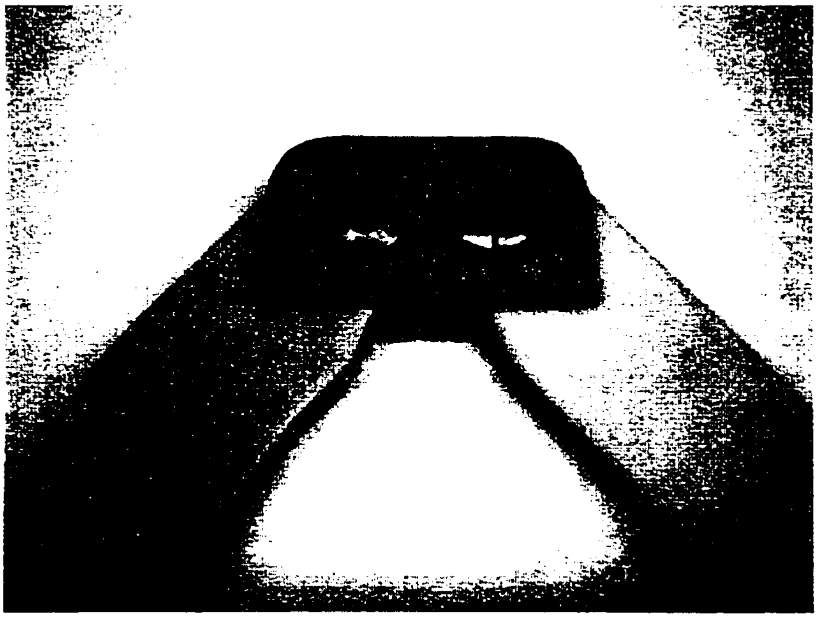


Figure 5.1 Picture of the knee brace

Common commercial plastics for this kind of product have a 30 percent content. It has been proven on the field that the knee brace works and it does not need any functional improvements. This central part holds the two axes of the geared arms of the knee brace. About 50 parts are typically needed for a production run. A picture of the original central part can be seen in Figure 5.2.

Figure 5.2 Picture of the original central part of the knee brace

When this research study began, the original mold was not available, and a reverse engineering study of the knee brace elements was done. All measurements were done either in the Browne and Sharp Coordinates Measuring Machine (CMM) at the Engel Laboratory or using a simple Vernier caliper. From these dimensions, a three-dimensional computer model was made, using Pro/Engineer, from Parametric Technology Corporation (PTC). Dimensions were rounded or adjusted keeping in mind that the part had been molded and had shrunk between 1% and 2.5%. In other cases, they were inferred according to the function of the part, as it is the case of the distance between centers of the pivot pins: This distance was derived from the known fact that it had be equal to the pitch diameter of a standard 13-teeth gear with an pressure angle of 20 degrees. A view of this 3-D solid model is shown in Figure 5.3.

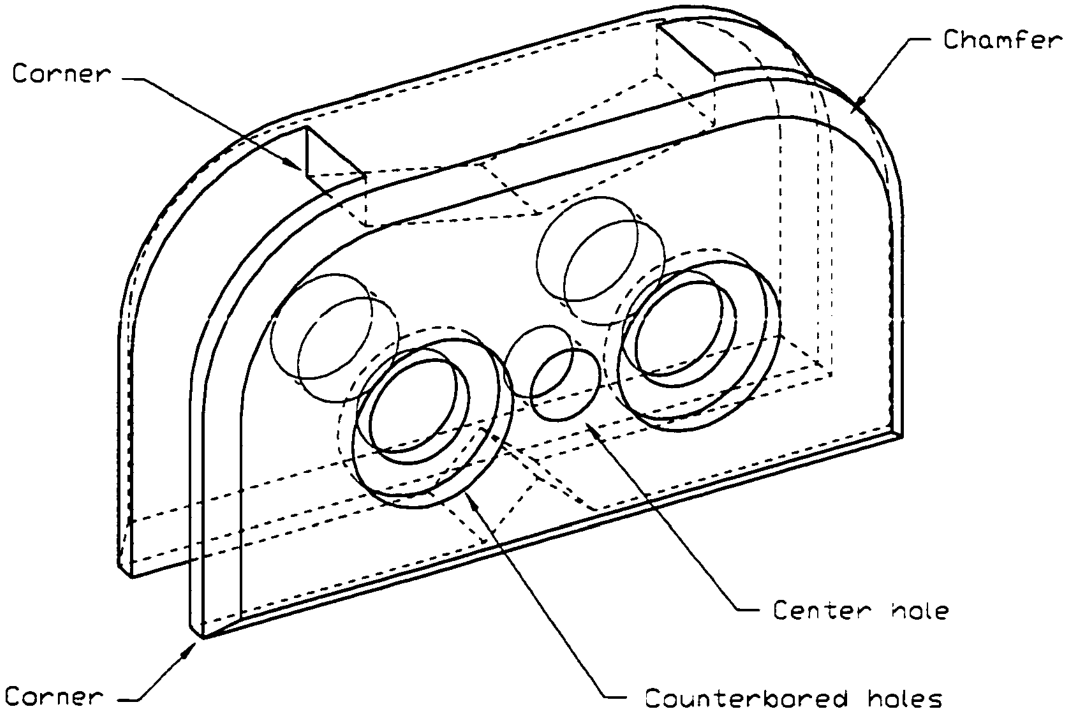


Figure 5.3 3-D solid model of the original central element of the knee brace

Redesign of the product for mold CNC machining

The basic geometric features of this particular object made it a perfect case for a CNC machined mold. Some details, however, needed to be analyzed in order to simplify even more the mold fabrication.

First modifications to the part: The internal details

From the analysis of this model, and from the observation of the actual parts, a 3-D model of the internal core needed to make the internal details of the part was made. To mold this part, two removable cores are needed. Because of the high stresses occurring in this element, the possibility of making it in two parts and then gluing them or welding them, to avoid having removable cores, was rejected. Usually removable core inserts are complex, and the system to remove them are even more complex. In SRP molding, the core inserts are removed by hand, and the design of the mold is simpler. In this particular case, the core inserts ended up being very simple.

The cores were made of a 1/4 inch thick aluminum plate, which is exactly the internal width on the part. The thickness of the "arms" of the knee brace is slightly smaller. To simplify the manufacture of the core, some details were modified so that it could be machined in one contouring operation. In this case, rule No. 4 was applied: Round the external corners of the part to a standard end mill radius. After it was determined that this change would not affect in any way the function of the part, the 3-D model was changed and the internal cores were machined. Because of the symmetry of the cores, both were machined at the same time, saving time and money. Two holes were drilled to hold them in a simple fixture. Two of these holes were cut off after trimming the parts to a specified length. The two remaining holes would have a locating function in the mold (see Figure 5.4).

Figure 5.4 Picture of the actual cores for the knee brace central component mold

Next modifications to the part: The external details

Then, the exterior of the part was analyzed and some details, that complicate or extend the manufacture of the mold, were found:

- The chamfer around the edges. It requires a special end mill. not available at this moment.
- The corners at 90 degrees. Usually inserts are required to produce sharp corners. which means more machining. and a more complex mold.
- The counterbored holes. These holes will contain hexagonal nuts which are a little bit larger than the holes themselves. When they are tightened. the corners of the hexagonal nut penetrate the plastic. preventing any rotation of the nut.
- A small hole in the middle. between the holes for the bolts. This hole apparently has no special function for the knee brace itself. It might be the resulting hole left by a locating pin for the core inserts in the mold.

The redesign processes

According to the author's theory. six quasi-concurrent processes happen in the mind of a mold designer when designing or redesigning a part for injection molding (see Figure 5.5):

- Positive thinking. "What does the detail look like in the part?"
- Reverse thinking: "What must the mold look like to make this detail?".
- Selection of a manufacturing process to make this detail on the mold. It also includes making the detail on the part. as a post-molding operation.
- Evaluation of the option. in terms of complexity. cost. and time.
- Modification of the detail. This includes the possibility to eliminate it.
- Analysis of the part detail in terms of aesthestics. function. and purpose.

It is the author's theory that these processes are quasi-concurrent. and that there is not any defined algorithm to describe this creative thinking process. Although some "jumps" from one process to another are more "natural" and "logical", they can be triggered by many other

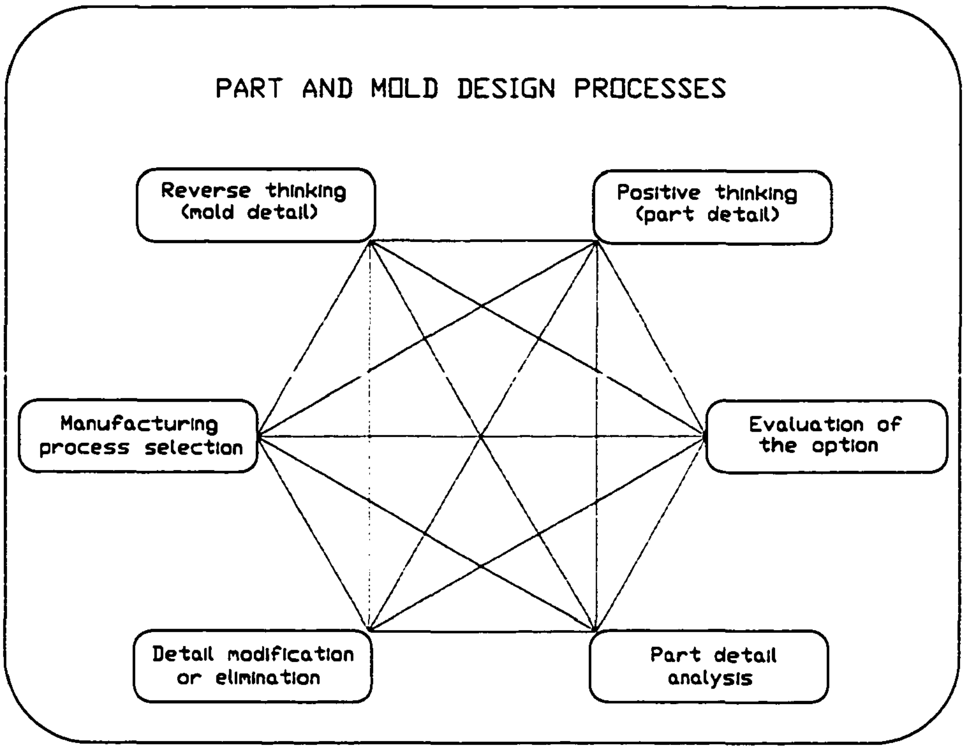


Figure 5.5 The part redesign thinking process for SRP injection molding

factors. These triggers may be the combination of thoughts and ideas previously found. new data. or no data at all. no solutions found. etc.

Modifications to the part

The results of this redesign process lead to the following modifications of the part:

- Two rounded corners for the internal details of the part. These rounded corners are generated by the radius of the end mill used to make the core inserts. as it was discussed previously.
- Four rounded corners for the lower external details.
- No chamfer on the part. to avoid using a special end mill to make this detail on the mold. It can easily be done by hand with a file or sandpaper. as a post-molding operation. It should be noted that often the original parts were sanded to round the chamfer even more.

- The two holes will be molded only on one side of the component. The other holes (in the back) can be drilled, using the molded holes as reference for the alignment. It was estimated that a simple wooden fixture could be built, and the 50 parts could be drilled in less than a half hour. Another advantage to drill the holes is that it increases the strength of the material by avoiding the weld lines in the material.
- Instead of the counterbored holes, two hexagonal pockets will be molded. The hole and the hexagonal pocket can easily be done by using a standard hexagonal bolt as a mold insert. Only the head of the bolt requires some machining. In this particular case, these bolts had been machined from hexagonal bar which was readily available.
- The small hole in the center was completely eliminated. It does not have any purpose for the new design.

The resulting 3-D model of the new component can be seen in Figure 5.6.

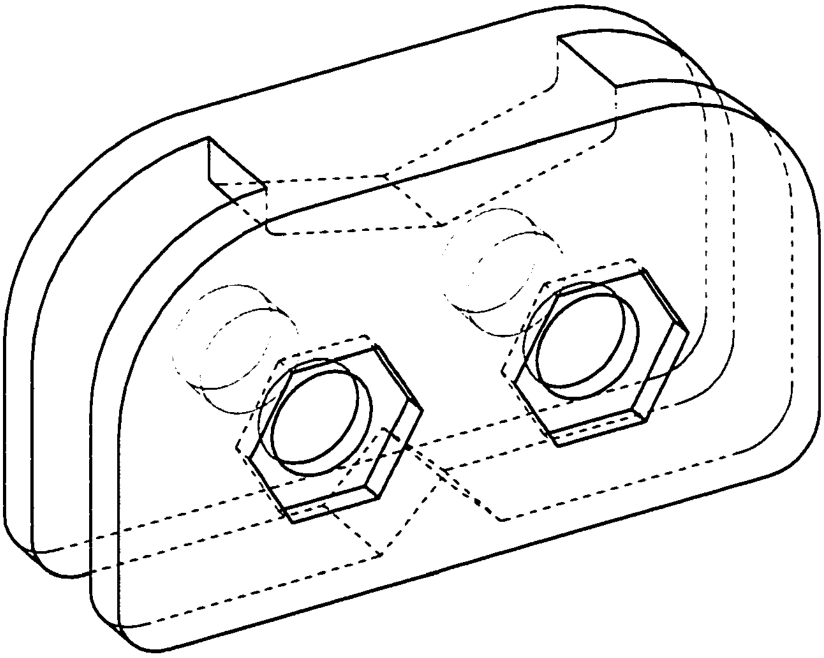


Figure 5.6 3-D solid model of the new central element of the knee brace

By performing this process, some details were lost, but others were gained. The rounded external corners, and the hexagonal pockets for the nuts are actual improvements to the part. What is lost is not considered to be worth the cost in time and money to make it as a detail in the mold. Other improvements to the part could have been made without complicating the fabrication of the mold, but it was not a goal of this study.

The design and construction of the mold

As it was discussed previously, the design of the mold and the redesign of the part are actually concurrent processes. One cannot exist without the other because of their interdependency.

The material chosen to make the mold is aluminum. Both, the cavity and core plates are to be made with standard 3" x 1" bar stock. A 6061-T6 aluminum was chosen because of its machinability, availability, weight, and high heat conductivity.

In this case, and because only 50 parts were needed, a manual molding process was defined. This would be a "class C" mold: It would be opened by hand on the workbench, not on the press. Therefore, only the rear mold half would require dovetail slots. These slots can be machined on any universal milling machine, with the head tilted 30 degrees, and with a standard end mill. The design of the slots is very robust. The depth of the slot is 0.173 inches. The distance from the bottom of the mold plate to the deepest part of the slot is 0.500 inches. Tolerances on both dimensions are +/- 0.020 inches.

Next, to mount the aluminum blocks on the machining fixture, two counterbored holes were drilled on both plates. The placement of the holes on the block, and the diameter and depth of the counterbored holes, require tolerances of +/- 0.020 inches. Since the holes on the block have to be aligned with the holes on the fixture, their relative placement is more important. The tolerance required in this case is +/- 0.003 inches which is easily obtained on any milling machine.

Next, both plates were mounted with cap screws on the fixturing plate, as it is illustrated on Figure 3.5. The CNC milling program for the rear mold plate will follow these steps:

- Center drilling of the dowel pin holes, the core insert holes, and the sprue well.
- Drilling of the dowel pin holes, and the core insert holes.
- Machining of the well, where the sprue will end.
- Machining of the top surface. This surface will become the parting line of the mold. The surface finish must be very good.
- Machining of the slot for the sliding core inserts.
- Machining of the pocket.
- Machining of the gate.

Note that no machining is needed for the lateral faces of the mold plates, since the reference will be given by the locating pins. The tolerances on the mold cavity and the alignment between the mold plates will be defined by the accuracy of the CNC milling center. Also, no vents were machined: it was assumed that the space between the sliding core inserts and the mold would work as a vent. The resulting rear mold plate obtained is shown in Figure 5.7.

The CNC milling program for the front half of the mold is very much similar, and will follow these steps:

- Center drilling of the dowel pin holes, and the sprue.
- Drilling of the dowel pin holes, and the sprue.
- Machining of the top surface.
- Machining of the pocket.

Note that the sprue will be straight and not tapered. It was assumed that because this would be a "class C" mold, the sprue would be removed by hand. The resulting front mold plate is shown in Figure 5.8.

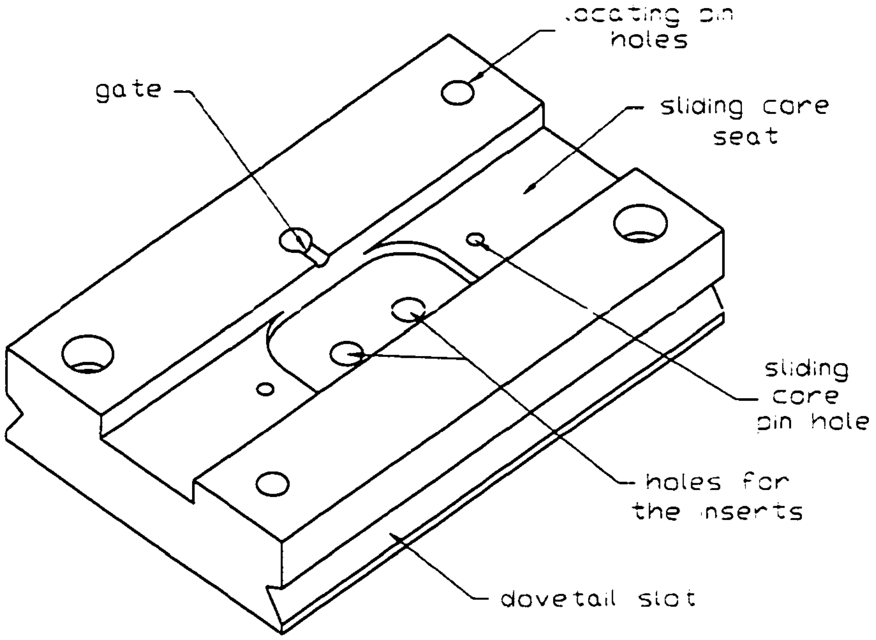


Figure 5.7 3-D model of the rear mold half .

The CNC program generated by a CAM package. The machining program was generated by EZ-Mill, a module from EZ-CAM, from Bridgeport Machines, Inc. Since the 3-D models were done in Pro/E, first a Pro/E drawing, at scale 1:1 was made. It was exported as a DXF file, one of the file formats accepted by EZ-CAM. The file was saved on a 3.5" floppy disk, and loaded on the computer connected to the VM40 Vertical Machining Center at the Engel Laboratory. This CAM package was used to define the different steps of the milling operation, based on the top view of the mold plate (see Figure 5.9).

Some finishing processes were done before the mold was ready to be assembled. The holes for the steel locating pins were reamed with an interference fit of 0.001 inch on the core plate, and a clearance fit of 0.001 inch on the cavity plate. The sharp corners were deburred by hand. Finally, the hexagonal core inserts were turned in a manual lathe. The pins and the hexagonal cores were assembled (see Figure 5.10).

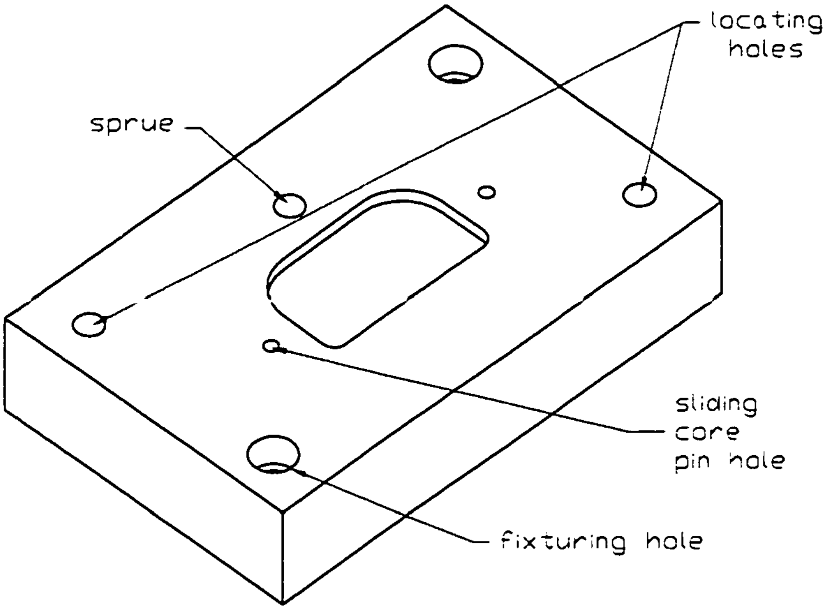


Figure 5.8 3-D solid model of the front mold half.

Experimental results

First mold assembly experiments

For the first molding experiments, the removable cores were held in place with steel pins going through the rear mold plate, through the cores, and into the front mold plate. It was expected that the molded part would stay on the rear mold plate, and to facilitate the removal of the part, a small slot, to fit a screw driver, was machined on the cores. The first mold assembly set up is shown in Figure 5.11.

The mold was then placed in the modular mold base, as it is shown in Figure 4.14. For the initial experiments, Polystyrene (PS) was used instead of glass fiber reinforced (GFR) Nylon. PS was chosen because it is easy to mold and does not require pre-dying of the resin (as Nylon does). PS was injected at 220 degrees Celsius and injection pressure of 17.2 MPa. The clamping force was the minimum allowed by the molding press: 68.24 kN. The resulting compression on the mold was around 11 MPa. The yield strength of aluminum 6061-T6 being

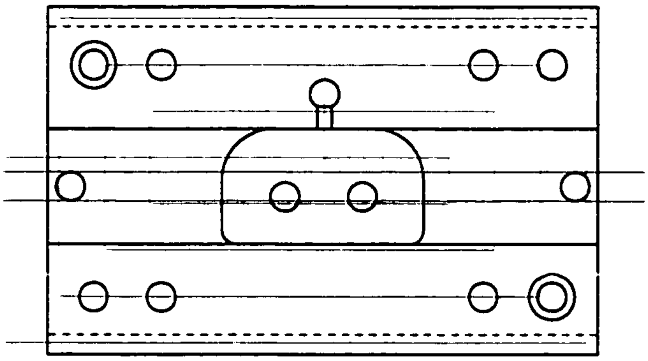


Figure 5.9 Top view of the rear mold used to generate the CNC program with EZ-Mill. The thin lines show the tool path for machining the top surface, the slot, and the gate.

275 MPa, no damage to the mold could be done by the clamping force.

Several parts were shot with this first mold. The first three were short, the shot size and the injection were increased and the fourth part was successfully injected. There were some problems to open the mold, and there was some flash in the middle section, between the two sliding cores. All the following parts had the same problem, but these first results were definitely encouraging. The main difficulty was to open the mold. The assembly setup for the sliding cores was definitively a problem. As the molten plastic was allowed to squeeze in between the cores, the injection pressure pushed the cores to the sides with a load estimated at 2700 N. This load was transmited to the steel pins, then to the mold plates. As the first parts had some sink marks on the internal surfaces, the injection pressure was increased to 25.8 MPa. The sink marks were less noticeable, but still there. The injection time was then increased from two seconds to five seconds and there were no more sink marks. By increasing the injection pressure, the opening of the mold became even more difficult. It was clear that another set up to hold the sliding cores had to be designed. Also, trying to pry open the mold on the slots was damaging the mold surface and it proved not to be a good way to open

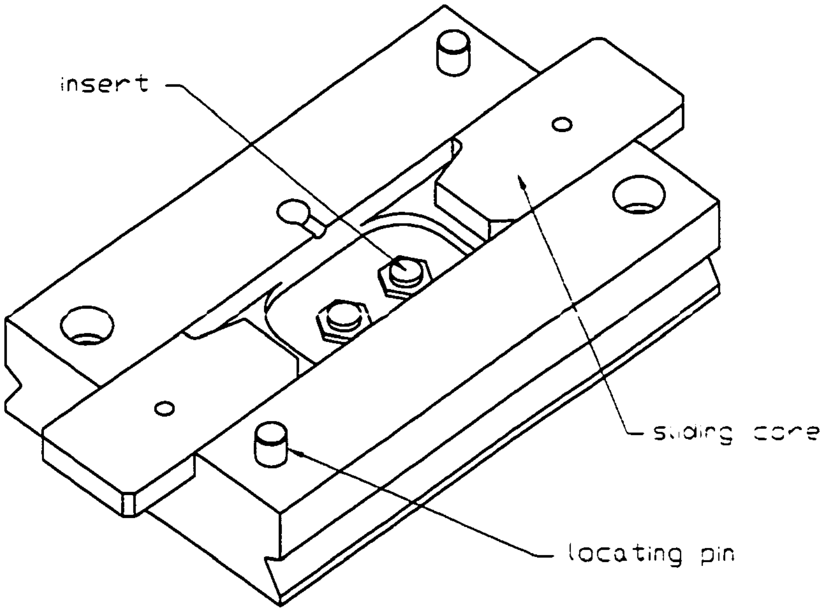


Figure 5.10 Rear mold half with the core inserts removed

the mold. Another problem was that one of the hexagonal cores had an undercut and stayed embedded in the part. That fact by itself was not a problem, but it was really hard to pull it from the part. This resulted in a little damage to the part.

Second mold assembly experiments

To solve the problems described previously, the hexagonal cores were extracted, refinished by hand and put back in the mold. To open the mold more easily, without damaging it, opening holes were drilled through the rear plate and partly into the front plate. In this way, the mold could be opened by knocking gently on pins placed in these holes. To keep the sliding cores in place, brass pins were made and inserted in the slot for the sliding cores. This new mold assembly setup is shown in Figure 5.12.

With this new mold assembly, a dozen PS parts were successfully made. Only a little bit of flash was still in the middle. In this case, when the mold base opened, the front mold plate

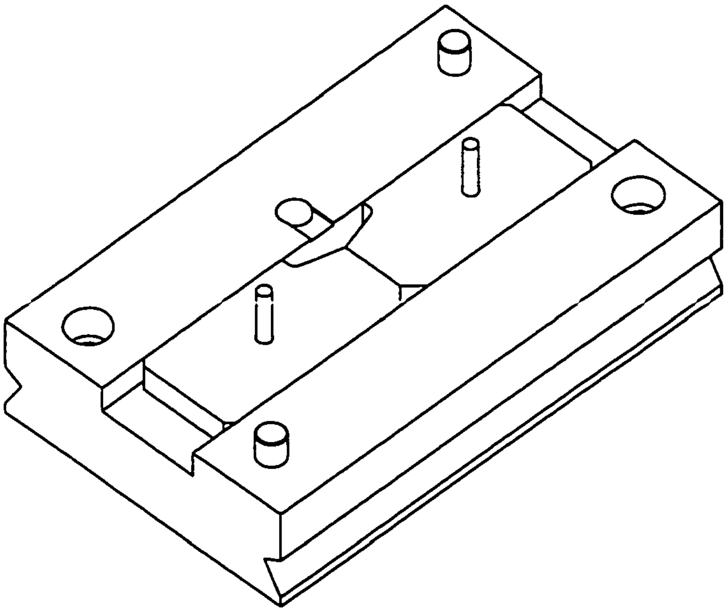


Figure 5.11 Rear mold half with the core inserts in place, as originally designed.

stayed on the front platen of the molding machine, attached by the sprue. To extract the part from the rear mold plate, one of the hexagonal cores was pushed from behind. The part was ejected with the sliding cores, which were then removed. The hexagonal core was easily pulled from the part, and placed back in its hole. The front mold plate was then pulled from the platen and the sprue was ejected with a pin and a hammer. At this point, the sliding cores began to get damaged by the pressure against the brass pin, and a small piece of aluminum sheet was used to keep the cores back into place

Then, another thermoplastic resin was used: Polypropylene (PP). Polypropylene was first injected with the same machine parameters as Polystyrene, except for the temperature. The first part was shot at 190 degrees Celsius. The first part was actually a mix of PP and PS that had remained in the screw. The temperature was definitively too low and weld marks were completely visible. When the part was extracted, the welds actually broke. Then the temperature was successively increased up to 260 degrees Celcius, the temperature recommended

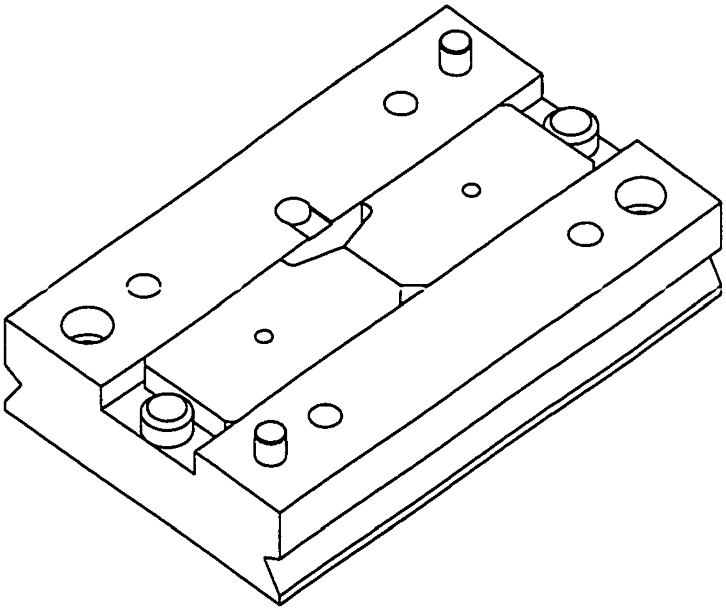


Figure 5.12 Second assembly setup for the removable cores.

by the injection machine manual. The increase in temperature effect on the part was clearly visible in the following parts. As the temperature increased, the weld lines were less visible, but sink marks on the rear mold plate were very noticeable. Also, with PP the flash in the middle was again present. Finally when the temperature of 260 degrees Celsius was reached, the injection pressure was increased to 77.3 MPa. This pressure is the median of the recommended range of injection pressures [50]. The clamping pressure, however, was not increased. The results were catastrophic for the mold. An analysis of the damage to the mold lead to these conclusions: As the injection pressure opened the clamped mold, the sliding cores moved, letting more plastic in between them. The projected area then increased even more pushing the cores against the tip ot the brass pins. Now the brass pin were like cantilever beams with a load at the end. The resulting force at the base of the pin was multiplied by this lever effect and the aluminum started to shear. Figures 5.13 and 5.14 show pictures of the damage caused to the aluminum block and the brass pin .

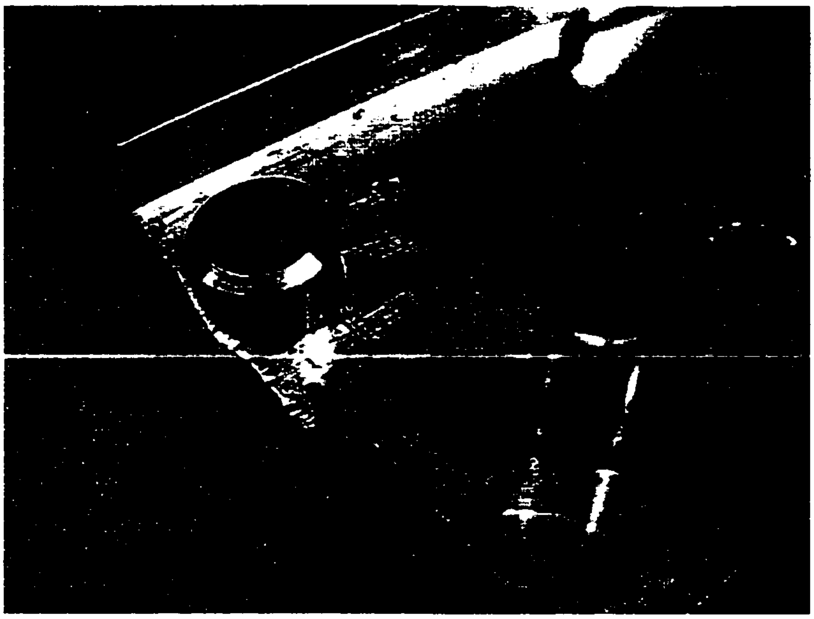


Figure 5.13 View of damage caused to the mold.

Third mold assembly experiments

After the that final shot, the mold was so damaged that it had to be either repared or a different mold had to be rebuilt from scratch. The latter option was chosen in order to prove that this rapid tooling approach worked well.

Only the rear mold plate was rebuilt with the following changes:

- The slot for the sliding cores was replaced by a pocket.
- The sliding cores were shortened to eliminate the damaged part and the slot used to extract them.
- Two holes to eject the cores and the part were added. The hexagonal cores would not be used as ejector pins anymore.
- The plate was longer, in order to facilitate the opening of the mold.

The rest of the mold design stayed basically the same. This new design was done by modifying the 3D computer model shown in Figure 5.15. As before, a top view drawing at scale 1:1 was made and then exported as a DXF file to be processed by the CAM software.

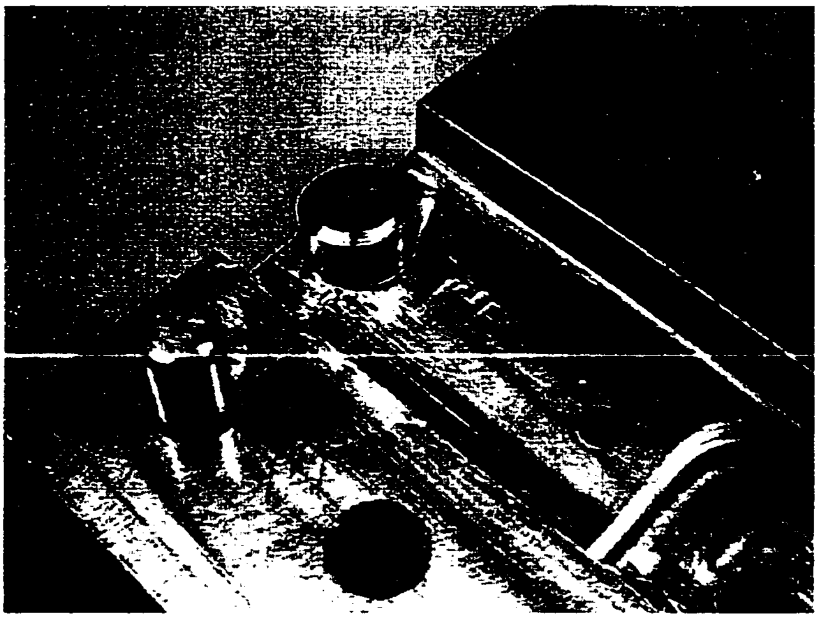


Figure 5.14 View of damage caused to the brass pin.

The resulting CNC part program, shown in Appendix C, was loaded on the CNC machining center, and the new mold was made. The holes for the locating pins and the hexagonal cores were reamed. The hexagonal cores and locating pins were assembled. The sliding cores were sanded by hand until a slight interference fit was obtained. This would ensure that no more flash would be present in the middle section of the component. It took less than four effective hours to rebuild the mold, from the 3D computer model changes to the fitting of the sliding cores. The resulting mold is shown in Figure 5.16.

Experimental results with the final mold

The new mold was placed in the mold base, and PS was injected again. The machine was set with the parameters that had yield good PS parts with the previous mold. The parts were all short shots. The injection pressure and injection time were increased, but with little improvement. Then, a vent was machined on the opposite side of the gate. In the previous design, no vents were needed because the air could escape freely through the sliding cores slot. This slot had been replaced by a pocket, and the air could no escape. The vent immediately allowed the plastic to fill the mold impression. Several different parameters were then changed

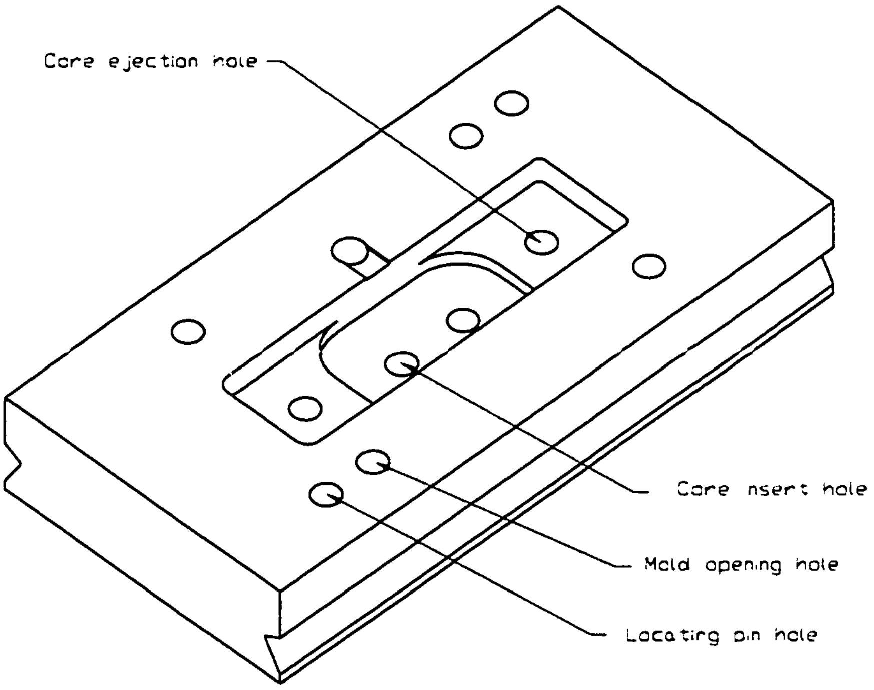


Figure 5.15 Third design version for the rear mold plate.

to see if the appearance of the part could be improved. Injection pressure and time were reduced to see if the flash on the sides of the mold could be eliminated or reduced. This actually lead to more short shots until an "optimal" part was obtained. By changing these parameters an important piece of information was also obtained: The location of the weld lines. If the part ever fails, it probably would occur there (see Figure 5.17).

The parameters were saved on the molding machine memory (program CHRIS01). The flash on the sides and the venting slot could not be eliminated but it can be easily removed. In Figure 5.18 a picture of a part, as it was extracted from the mold, shows the flash on the sides and at the vent.

Several of these parts were finished by removing the flash and drilling the holes. In Figure 5.19, a knee brace was assembled with one of these new central components. It fit perfectly and even showed an unexpected good behavior to bending, considering that Polystyrene has a relatively low strength.

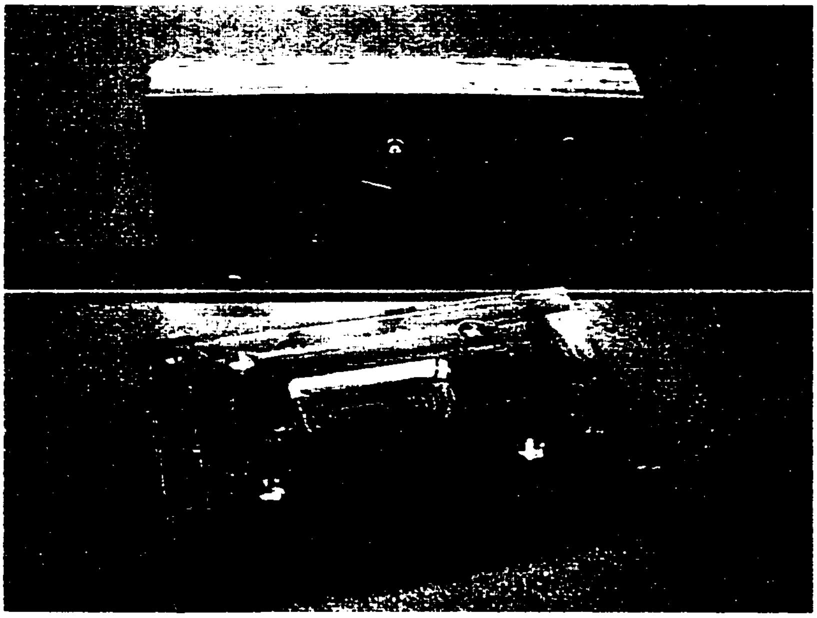


Figure 5.16 Picture of the third version of the mold.

Figure 5.17 A short shot showing the location of the weld lines.

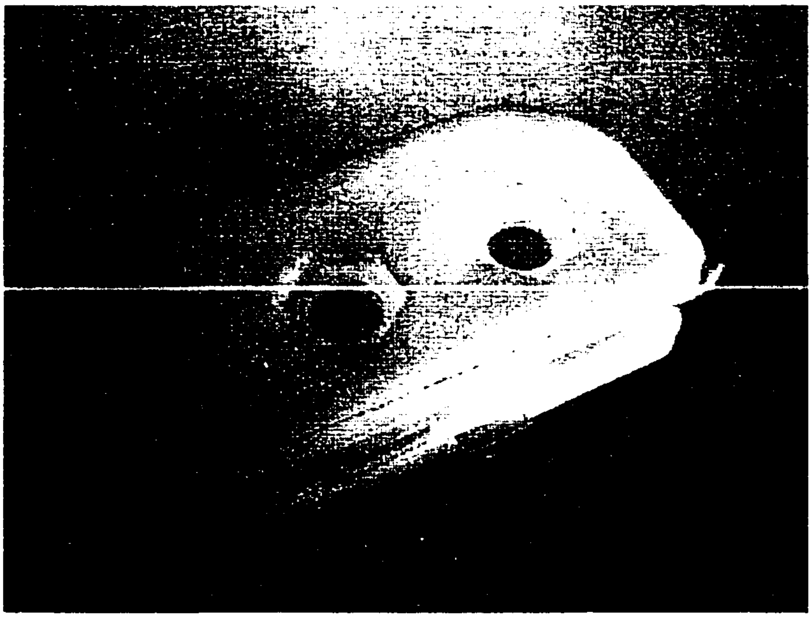


Figure 5.18 New central component, as extracted from the SRP mold.

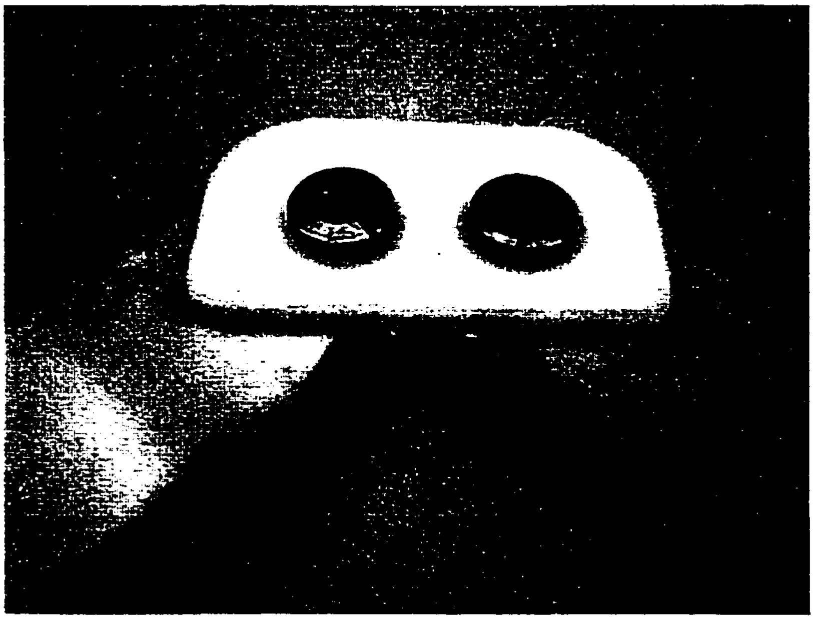


Figure 5.19 Knee brace assembled with the new central component.

6 HYBRID MACHINED/CAST MOLD DESIGN

In this chapter, a prototype mold will be built and tested. The design of this mold follows the rules and guidelines defined in chapter 3, "Design Rules for Short-Run Production Injection Molds". In particular, the fourth principle, which states that a mold hard to machine should be cast, will be applied. In this experimental approach, only the details that are hard to machine will be cast. The injection molding process, the prototype parts, and the prototype molds will be evaluated.

A study case: knee brace arm component

In this study case, the part to be molded is the geared arm element of the knee brace described in the previous chapter. The material used in the original part is the same glass fiber reinforced (GFR) polyamide. The part does not need any functional improvements, and about 50 parts are needed. It was known that the part was originally injected in a flat parting-line mold, and then was reheated and bent in a special die to make "right" and "left" arms. The plastic has to be injected from one end to avoid a butt weld line. In the case of Nylon 6-6 with 30% GFR, the tensile strength retention value is in the range of 56 to 64%. This low value is mainly due to the fiber orientation at the butt weld line [29]. The body of the arm is very simple, and can be molded in a machined pocket. The main problem in this case is the geared part of the arm (see Figure 6.1). These details are very hard to machine on a mold. It was assumed that the original mold had an insert that had been either hobbed, or eroded by an electric discharge machine (EDM). Standard tools exist to make the gear teeth, but not to make the mold of a gear tooth.

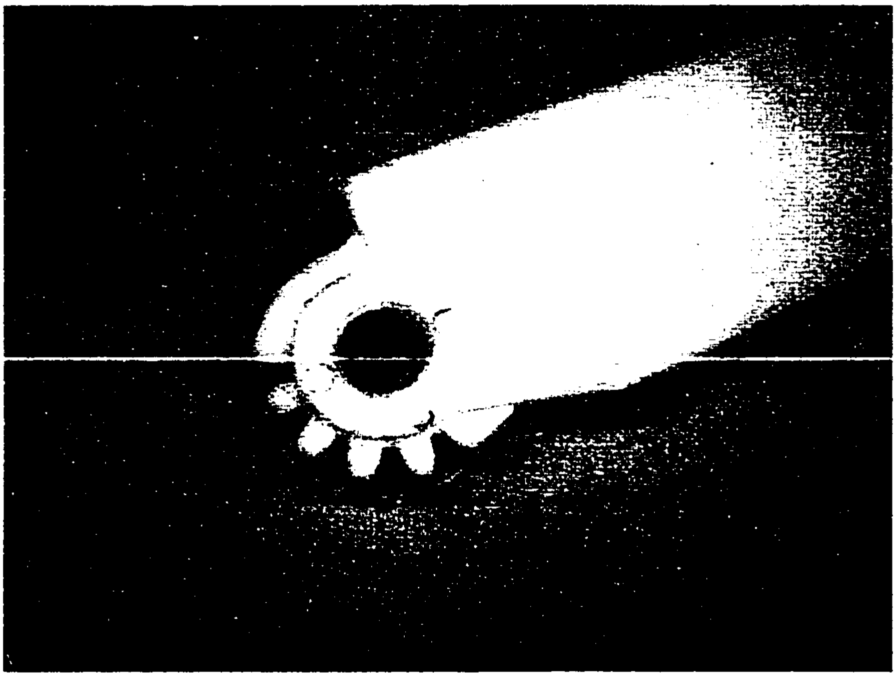


Figure 6.1 Picture of the gear detail on the leg.

Design of the hybrid machined/cast mold

This study case was ideal for researching about hybrid machined/cast (HMC) molds. The idea behind HMC molds is to shorten the total fabrication time of the mold by combining machining and casting. Some parts have details that are hard to reproduce by machining the mold directly. In these cases, a pattern can be made, using some additive or subtractive RP method, and a mold insert is cast. This approach is not new for cast inserts that reproduce the whole part, but it is for reproducing only some details of the part. In this specific case, the cast inserts to be researched are made of thermoset resins, and will be directly cast into the mold.

Design of the mold for the knee-brace geared arm

As in the previous study case, 3D computer models of the part and of the mold were made and analyzed, together with the original geared arms. By studying the flow lines in the plastic is was determined that the original mold had one gate at the opposite side of the gear. The hole for the gear was also molded. As the ejector pins marks indicate, it was probably very hard to extract the part on that end of the mold.

It was decided that for the redesigned mold the gate would be placed in the same position. Also, the hole would not be molded, but drilled as a post molding operation. The mold would be made of 6061-T6 aluminum bar stock (3" x 1"). The mold would extend beyond the edges of the mold base, but only 1.5 inches on each side. There would not be any interference with the functioning of the mold press. The 3D solid model of the rear mold plate (Figure 6.2) shows that all the details of the mold can actually be made on that half of the mold. The front half would only have the sprue, and maybe a half runner.

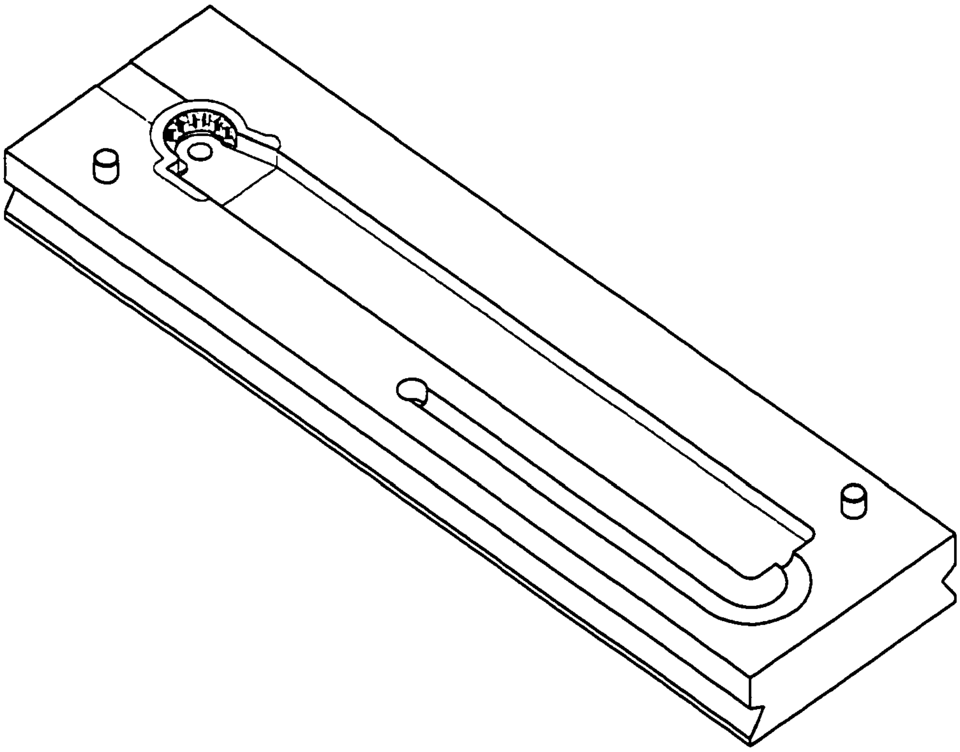


Figure 6.2 3D computer model of the knee-brace arm mold.

For the cast gear detail, a pocket would be machined, leaving enough space for the casting material (about 0.25 inches). Then, using an original geared arm as a pattern, the epoxy would be cast inside that pocket. Figure 6.3 shows the gear detail in the mold. An ejector pin could later be placed in the center of the gear detail to facilitate the extraction of the part.

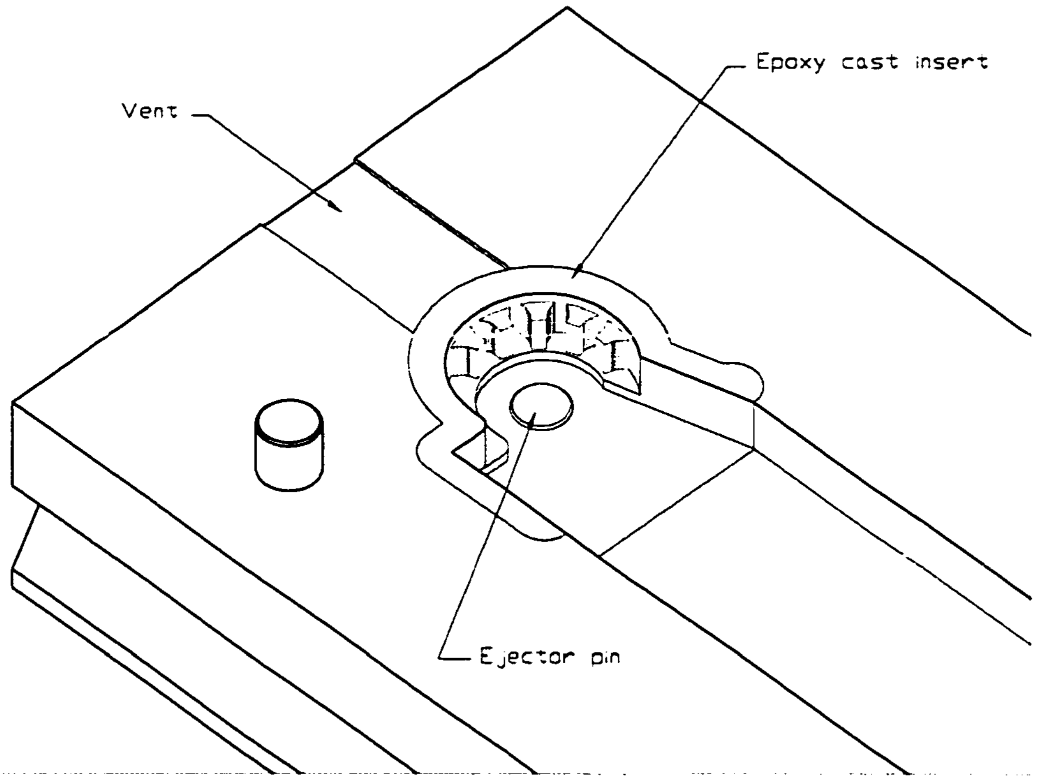


Figure 6.3 Detail of gear in knee-brace arm mold.

Experimental results

The mold was then machined. using the same CAD/CAM procedures described in the previous chapter. The cast inserts. however were not made on that mold. Since it was an experimental procedure and the knee-brace arm is a long part and therefore a lot of plastic could be wasted. a shorter version of the mold was made for the experiments.

The experimental mold for HMC tooling

This smaller mold would actually have a pocket in which different mold inserts could be placed. The front mold plate is just flat. with a sprue gate. The front plate is reversible: The sprue can feed the resin either on the cast epoxy side. or on the machined aluminum side. A picture of this mold is shown in Figure 6.4.

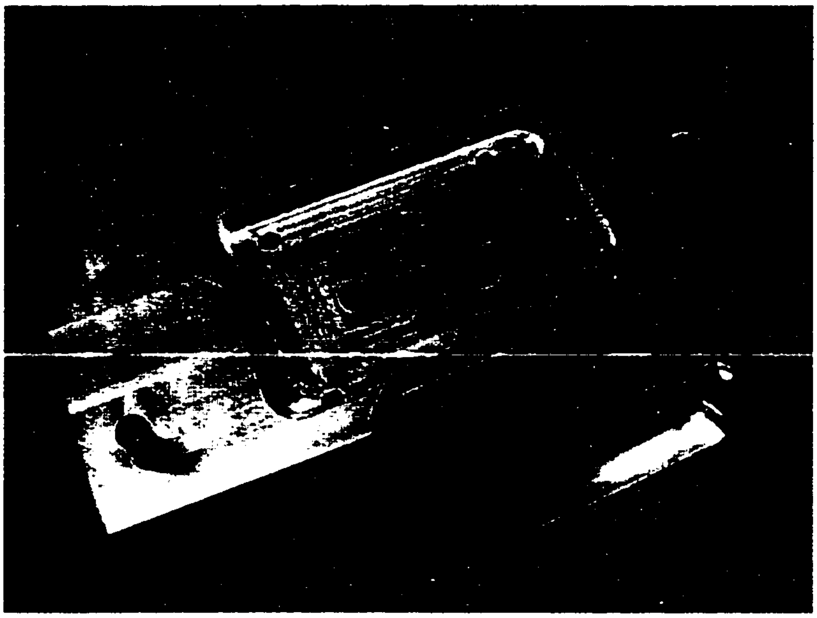


Figure 6.4 Mold with the clear epoxy mold insert.

The mold inserts are machined with a small interference fit, and sanded by hand.

Experimental results with paste epoxy mold insert

The first mold insert was made with a simplified pocket for the cast material. Common materials that can be obtained at any store were chosen to do the first experiments. The first experimental material was "Plug-N Patch", from Power Poxy, Inc., an epoxy/amine paste mix. This product has to be mixed, applied after one minute, and sets in a half hour. The product is completely cured after one hour. It seemed to be an ideal product for rapid tooling applications.

The aluminum insert was prepared: All surfaces were thoroughly cleaned first, then the surfaces which had to be demolded later were sprayed with silicone mold release [1]. The geared arm pattern was placed in position, and all was held together by a back plate and "C" vises. The casting material was prepared according to the instructions, and applied into the cavity. It stayed overnight like that, and in the morning the pattern was extracted. The back was then machined to remove the excess material. The resulting mold insert is shown in Figure 6.5.

---

[1] "Slide Paintable Mold Release", from Slide Products, Inc.

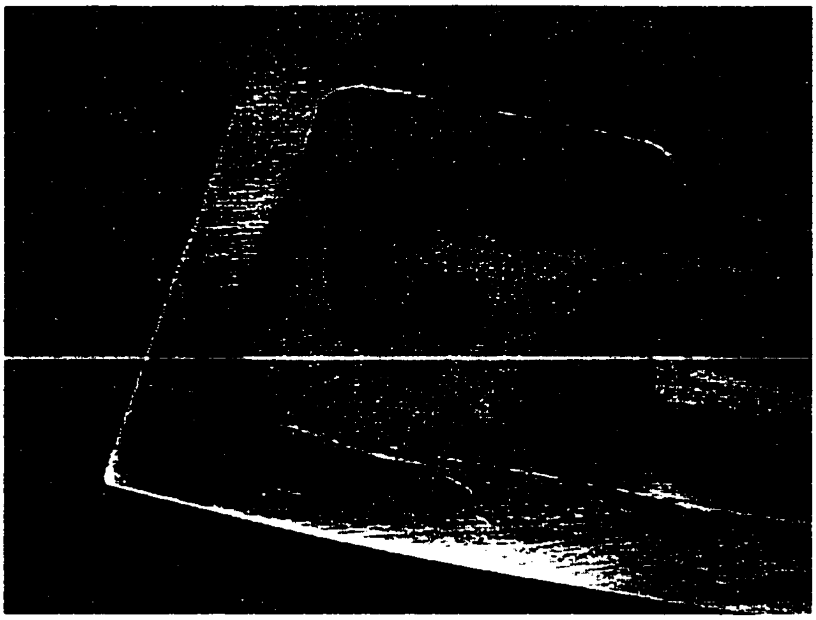


Figure 6.5 Paste epoxy mold insert.

The appearance was generally good, but some details had not been completely reproduced. It was clear at that point that by just pressing the epoxy paste against the pattern with the fingers would not always reproduce the fine details. The insert was not tested for molding. To reproduce these fine details, a liquid epoxy was then tested.

Fabrication of a clear epoxy mold insert

Again, another commercial product was used: "2-Ton Crystal Clear Epoxy", made by Devcon Consumer Products. A new aluminum mold insert was machined. Then, it was prepared to cast the clear epoxy, as described previously. The epoxy was mixed thoroughly and poured slowly into the cavity. Small bubbles (formed while mixing the epoxy) were hard to remove and the product was setting fast. All but one of largest bubbles were removed with a thin copper wire. Although this bubble was a mistake, it provided some valuable information on later tests. This product sets completely in about two hours, and is completely cured in eight. The insert stayed overnight in a warm, turned off, oven.

The pattern was very easily removed. The back of the insert was machined to remove the excess material. The large bubble, close to the ejector pin, is clearly visible in Figure 6.6.

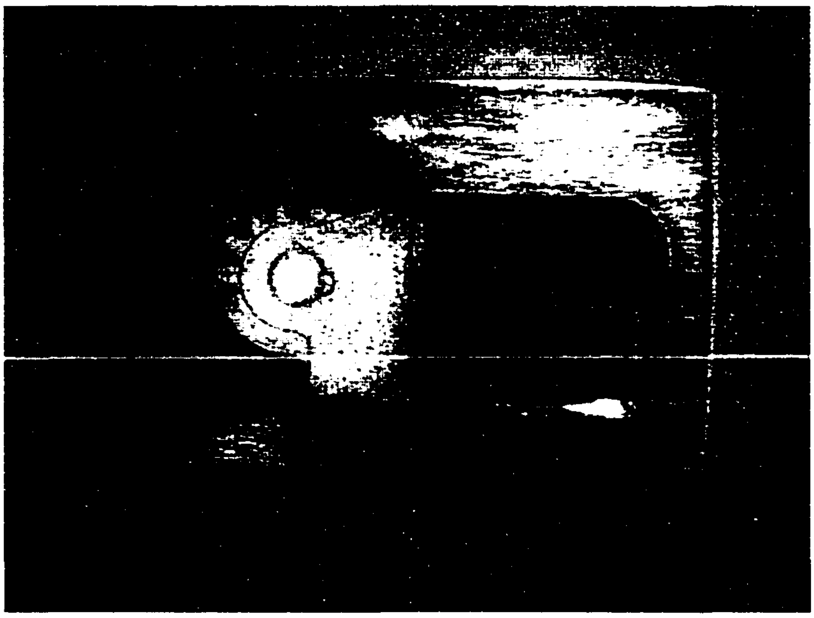


Figure 6.6 Clear epoxy mold insert.

The mold insert was then sanded and placed in the mold pocket. The mold was located in the mold base with the dovetail clamps. Finally, the mold base was placed in the molding machine.

First set of results with a clear epoxy mold insert

The mold was sprayed with silicone mold release and the first parts were injected with Polystyrene (PS). The first observations were that the mold had good venting (there was some flash around the gear end of the mold to prove that). The insert height was a little bit smaller than the mold insert depth for that purpose. The surface finish of the part, where the mold was machined, was very smooth. The surface finish of the part, where the mold was cast was very rough. All the small bubbles that were close to the surface were burst by the injection pressure. Molten plastic penetrated in them, and after solidifying, left this aspect of small spheres on the surface of the part. It is interesting to note that the epoxy used has a glass temperature lower than the injection temperature of PS, and that the plastic spheres were "extracted" from the bubbles without breaking them. This phenomenon is particularly clear for the big bubble, and lasted until the eighth shot, when it cracked. Besides that, a big sink mark appeared on the top surface of the part and smaller ones between the gear teeth. This corresponds to the bottom part of the mold, which was the last to solidify. Although ejector pins had been included in the mold, they were not necessary. The part was extracted from the mold when the mold base was opened. The part always tended to stay attached to the sprue, which stayed on the front mold base. A picture of one of these parts can be seen in Figure 6.7.

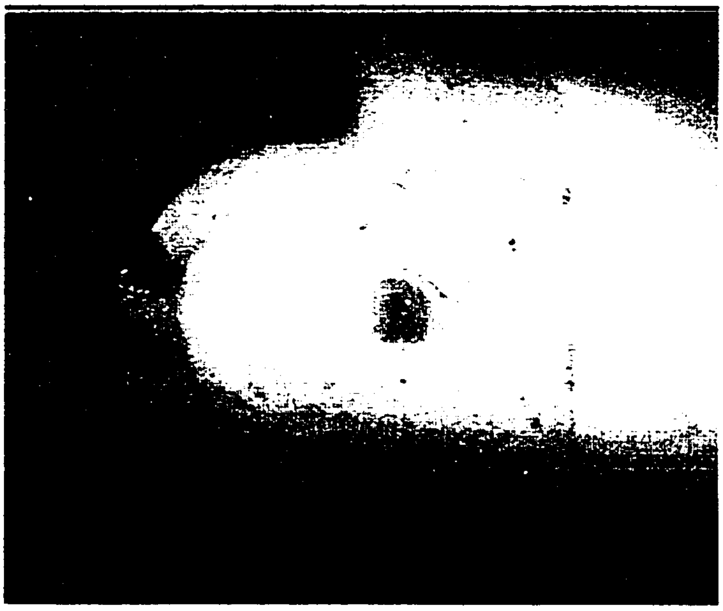


Figure 6.7 Geared arm molded with gate on the machined aluminum side of the mold.

It was obvious that the plastic tended to solidify last at the cast insert end, which caused the sink marks problem. Several other parts were shot trying to increase the holding pressure and the holding time, without any real improvement. However, the sink marks tended to change: Instead of being wide and shallow, they started to become narrow and deep. The extreme case happened for the tenth shot. This phenomenon was attributed to the fact that no more silicone release had been applied. This was confirmed after more release was sprayed on the mold for the 14th shot: the original sink mark was there again. The sink marks were just moving from the surface, to the interior of the part. This was confirmed later by cutting the 12th part: Void holes were found at the cast insert mold end.

During the ninth shot, and for no apparent reason, the mold did not close completely and the part had a lot of flash. The mold had to be opened by hand. Plastic had penetrated in every single space of the mold. Although it was not noticed at the moment. it is believed that the mold insert was cracked during this operation. The next shots would have "flash" where the crack was. But it was just a small crack and the mold was still good.

After the eleventh shot. the tip of the gear teeth started to be incomplete for no apparent reason.

Second set of results with a clear epoxy mold insert

After the 18th shot. the mold setup was changed. The front mold plate was turned: The sprue gate was now on the cast insert end. The mold had to be relocated. to align it with the sprue on the mold base. It was assumed that the sink marks would now be eliminated since hot melt would be available through the sprue.

The first part that was obtained with this setup showed that the sink marks problem was almost solved. There were still some small sink marks between the gear teeth. But in general. the part was better: Every small detail had been reproduced adequately. A picture of this part is shown in Figure 6.8.

This was the only part that could be made. During the opening of the mold. the cast insert was pulled out of the aluminum pocket. When the insert was pushed back into place. the bottom part had not been properly engaged and was bent. The epoxy insert at that point was soft and flexible. By the time the insert was pushed out again to engage it properly. the epoxy had become rigid and fragile. When the insert was pushed in again. it cracked. The mold insert was then extracted from the mold. A picture of the cracked insert is shown in Figure 6.9.

Third set of results with an aluminum filled epoxy mold insert

From the previous experience. the mold insert was modified so that the epoxy insert would not come out. And another commercial product was tested: "Poxy Weld". from Power Poxy

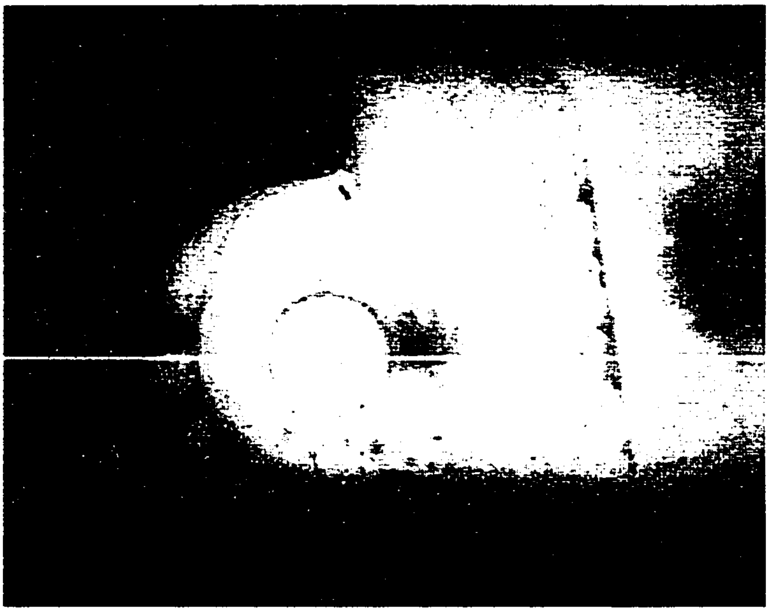


Figure 6.8 Geared leg molded with a sprue gate on the epoxy insert side of the mold.

Adhesives, Inc. This product is an epoxy/amine compound that contains aluminum powder and Kevlar fibers. Aluminum filled epoxy (AFE) is definitely a better heat conductor than epoxy alone. Its abrasion resistance is also improved [11]. The difference in heat conduction between the aluminum and the AFE being smaller, the sink marks should be less.

The mold insert was prepared, and the AFE was mixed and cast. It was left overnight to cure, as recommended in the instructions. When the pattern was removed, part of the cast insert broke. Not enough silicone mold release had been used, and part of the AFE bonded to the surface of the gear teeth. Also, some bubbles where visible, in between the gear teeth, which caused severe undercuts in the mold. The mold insert was tested anyway, for testing strength and heat conduction behavior.

Twenty parts were made with a sprue gate on the mold insert side of the mold. The first parts showed that the details were adequately reproduced. As in the clear epoxy mold, a change in the texture was visible, but less rough. There were less "spheres" at the surface. Only one small sink mark was visible, between two gear teeth, in shots one, two, three, and six.

Figure 6.9 Fracture of the clear epoxy mold insert.

The undercuts, produced by the bubbles during the casting of the insert, ended up breaking the gear teeth details. The fifth, seventh, and sixteenth shots in particular sheared relatively big pieces of the insert details.

The parts were easily removed from the mold. They actually stayed on the front mold plate. The cooling time was 30 seconds. The total cycle time, including the manual operations, was under 90 seconds.

Results with a fast-cure aluminum filled epoxy mold insert

The next experiment was designed to solve three problems encountered with the previous materials. It was clear that heat transfer of the insert had to be increased if an aluminum mold was used. Also it was clear that the strength of the material had to be improved. Aluminum, as a filler, could solve that problem, but the third problem remained: The surface finish, was not good enough. The cause of that: bubbles remainining in the cast insert. Most of the bubbles due to air trapped in corners and small details could be removed by using a thin copper wire when clear epoxy was used. With AFE, however, it was harder to know if the bubbles had been removed, or were still there. To solve this problem, two possible options were available:

Use a vaccum chamber to "pull" the bubbles out, or lower the viscosity of the epoxy by heating it. Since there was an oven that was readily available at the Engel Lab., the latter was chosen. The epoxy viscosity would be decreased by pre-heating the resin and catalyzer, and the mold. Time was critical since at high temperature, the epoxy cures faster. The epoxy chosen was "5-Minute Fast Drying Epoxy", made by Devcon Consumer Products. The double-syringe with the two-part epoxy was put in the oven at a temperature of 160 degrees Farenheit, for 45 minutes. Then, each part was mixed with aluminum powder. The proportional volume of aluminum was approximately 40%. Then, both parts were mixed for a minute and poured into the mold. Due to its low viscosity, the epoxy was easier to cast. The material was filling the details fast. Small bubbles were breaking at the surface of the liquid epoxy. Less than five minutes later, the epoxy was solid. The mold was put back in the oven. The oven was turned off. An hour later, the pattern was removed: All details were complete, the surface finish was better. Parts began to be shot just an hour and a half after the insert had been cast.

Fifty parts were shot, with a sprue gate on the insert side of the mold. The parts had a relatively good surface finish and with no visible sink marks. The average cycle time was 84 seconds. Basically, the 50th part looks just as good as the first. There was flash on all the parts: This was due to the thickness of the aluminum insert used to cast the AFE. It had been used several times, and each time the back had been machined a few thousandths of an inch to level the surface of the cast epoxy. Later shots were tried at different injection pressures and shot sizes to reduce the flash. It got better, but was never completely eliminated.

Although some problems remained to be solved, it was shown that epoxy cast inserts can be used to make rapid tooling. A theoretical analysis of the thermal conductivity problems found in hybrid-machined/cast molds is developed in Appendix D.

7 CONCLUSIONS

Overall, it can be said that the experimental results were encouraging. A lot of data and ideas were generated during this experimental research study and some of them should be investigated more deeply. Some of these ideas are very promising and could lead to innovative ways to do "flexible" injection molding.

Mold base for flexible fixturing of mold plates

Overall, the mold base design proved to be excellent and worked adequately for all the molds that were used with it. A few improvements can be accomplished to extend the capabilities and flexibility of the mold base.

The dovetail clamps worked just fine, and they were originally designed so they could be used for both semi-automatic molding, and for manual molding (that is, manually extracting the mold from the mold base after each shot). The advantage of the dovetail clamp is that it can guide, and also rigidly hold the mold plates, because of its configuration (the 60 degree angle). In manual molding however, it took a relatively long time for the setup of the mold plates on the mold base. It was hard to find the "right" position of the clamp for an easy engaging of the plate in the slot, without too much free play. This problem can be solved by having a "T" clamp. The "T" slot formed by these "T" clamps is easier to adjust for guiding the mold plates, without clamping them. The corresponding machining on the mold plates would also be simplified.

Another improvement could be to have a more flexible way to eject the parts. Although the current mold base provides for "normal ejection", it is quite restrictive on the possible positions of the ejector pins in the mold. Modular independent ejection units could be used for that purpose.

The alignment of the sprue, although simple, was a relatively long process. By using a retractable locating pin on the rear mold base, and a locating hole on the back of the rear mold plate, this problem could be solved easily.

The current mold base must be seen just as a prototype, and an instrument to generate new ideas for designing mold bases for flexible fixturing of mold plates. The ideal flexible mold base would automatically recognize the mold plate, locate it, and clamp it. But any other improvement, like having a faster clamping system, would increase the throughput of SRP molding.

Design for rapid CNC mold machining

Design for SRP mold machining should be seen as just another subset of rules and guidelines of Design for Manufacturability. As these rules are used to make molds, and new problems are encountered, new rules can be defined.

The trend to produce less parts in injection molding, could lead to a different way of designing molded part. It can also lead to combine more molding processes with CNC machining post-processes.

Usually post-molding processes are regarded as being a disadvantage. In this case, it was found that it can have an added value by improving the strength of parts by avoiding the "inevitable" weld lines.

Research in modular fixturing for post-molding processes can be a new and promising field, as post-molding processes become more common for SRP injection molding.

In this research, "normal" milling speeds and 2D CNC software were used. Literally everyday new CAM software improvements allow for more complex and sophisticated milling operations, in more than three axis. This, combined with new milling tools and CNC milling machines technologies will allow for machining aluminum at speeds well above the ones used for the research. This will close the gap between subtractive and additive rapid prototyping.

As it was discussed in chapter 2, five-axis high-speed milling can sometimes still be very effective and surpass other rapid tooling technologies. These kind of machining centers are also getting more sophisticated and capable, and as new software can assist the programmer, it will become also more common.

Design of hybrid machined/cast molds

From the results obtained in this investigation, it is clear that the thermal conductivity of the cast insert and the mold material should be similar. This can be achieved by either increasing the conductivity of the cast insert material (with more aluminum powder), or by using another material for the mold (epoxy, polyurethane, etc.)

As the cast epoxy inserts gets hot during the injection molding process, it reaches its glass temperature and becomes soft. Proper backing of the insert is necessary because of the high pressures in the cavity. If not, the molten plastic will push or deflect the soft insert, leading to low dimensional tolerances and/or breaking the insert.

Another point with respect to HMC molds is that it does not have to be restricted to machining a mold and casting the inserts. This process could be repeated: Like this, the final mold could be the result of successive operations of casting and machining, similar to the Mcubed or SDM RP technologies.

Another improvement to the HMC mold manufacturing would be to inject the inserts at high pressure. With this, paste epoxy would probably be more effective than liquid epoxy, and easier to handle. Also, higher metal content could be achieved in metal filled epoxy.

New composite boards, especially designed for high-speed milling, are being invented. Combining high-speed milling with HMC mold making can lead to new frontiers in mold making technology.

Future research

This research study lead the author to think of many other rapid tooling processes that can be studied and developed. Some of these are summarized below:

Flexible particulate bed mold. This mold making concept utilizes an RP master pattern to shape a particulate bed mold impression. A silicone or latex membrane would separate the particulate bed and the pattern (or the molten resin, when injected). The metallic powder could be poured, compressed, and then vaccumed to hold the particules in place when the pattern is removed. This mold could be the ultimate flexible mold: Several patterns could be made in one mold, and the process could be automated.

The "One-shot" mold. This mold would use a metal filled wax insert. The insert would be made by injecting the slurry between the cavity pocket and a modified master pattern. The mold making process would be similar to the ceramics injection molding process described in chapter 2. The mold would last for one shot at the least and perhaps two or three at the most, but it is 100% recyclable and it can be rebuilt in a matter of minutes.

Layered Mold Machining (LMM). Layered mold machining, a technique invented by the author, in which the mold is made of several plates and the machining only involves contouring operations could become a standard way of fabricating specific molds in the future. Although it can already be used by ball-end milling the boundary surfaces of the layers in 2 1/2-axis CNC milling, it could become even more attractive with 5-axis CNC machining. Adaptive slicing procedures are investigated by Hope, Roth and Jacobs [51] for improving the geometric accuracy of layered manufacturing techniques would shorten even more the machining process. In this study, they use sloping boundary surfaces that match closely the shape of the required surface. By using this methods, they are able to eliminate the stair case effect, typical in layered manufacturing. Thicker layers are also possible, without compromising the surface tolerances.

APPENDIX A COMPUTER SPREADSHEET FOR COST ANALYSIS

The cost analysis for an SRP mold can be done with a simple program, or a computer spreadsheet. This kind of analysis must stay as simple as possible in order to be efficient. A lengthy cost analysis, with too many variables, might end up delaying the actual fabrication of the tool.

The following cost analysis example is done for a short-run production (50 parts). **The first approach** is to make an SRP mold with all the details, and a simple ejector system to be able to use the injection molder in automatic mode. The ejected part does not need any finishing, except the trimming of the gate. **The second approach** is also an SRP mold, but in this case, no ejection system is included. Also, the molded part will need simple finishing operations (drill two holes and chamfer some edges with a sander).

The cost of one finished part is: **Uc = Tc/N + Mc + Fc + Rc**

where:

Uc = Unitary cost
Tc = Tool cost
Mc = Molding process cost per part
Fc = Finishing operations cost per part
Rc = Resin cost per part
N = Number of parts

- The cost of the tool (Tc) can be estimated as the time required to machine the mold.

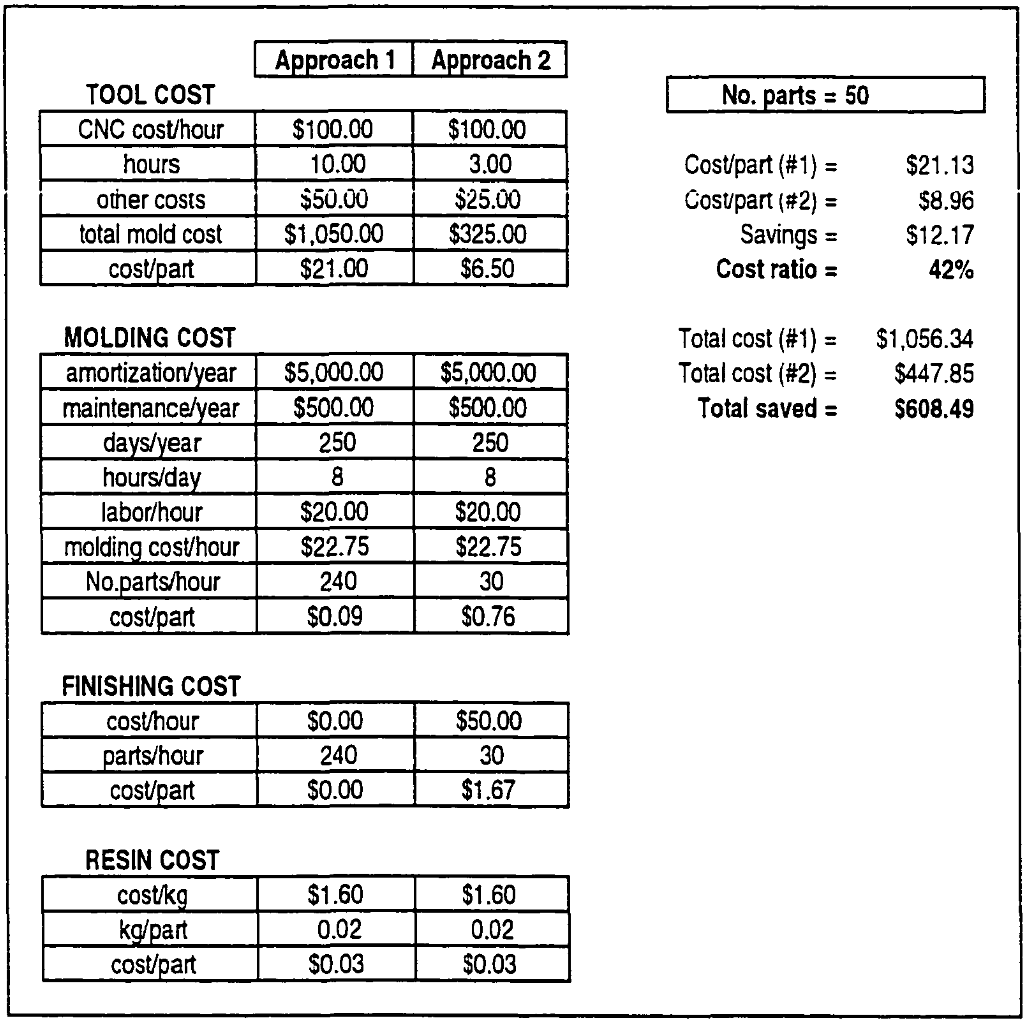
multiplied by the hourly cost of a CNC machine ($100/hour in this case).

- The molding cost (Mc) can be estimated as the operating cost of the molding machine (including hourly costs of amortization, maintenance, and labor) divided by the number of parts per hour.
- The finishing cost (Fc) can be roughly estimated as the labor cost per hour divided by the number of parts per hour.
- The resin cost (Rc) is basically the cost of the polymeric resin per kilogram, multiplied by the weight of the part (including sprue and runners).

In Figure A.1, the cost analysis is shown for 50 parts. In this case, the second approach cost is approximately 32% of the first approach cost. The total amount saved is $608.49.

In Figure A.2, a spreadsheet using the same equations is done for a number of parts ranging from 20 to 500. From the table it can be determined that both costs will be similar when the number of parts is roughly 300.

In Figure A.3, the spreadsheet data is shown graphically. This is the typical behavior of the unitary cost of a molded part, using two different approaches for making the tool.



TOOL COST

	Approach 1	Approach 2
CNC cost/hour	$100.00	$100.00
hours	10.00	3.00
other costs	$50.00	$25.00
total mold cost	$1,050.00	$325.00
cost/part	$21.00	$6.50

MOLDING COST

	Approach 1	Approach 2
amortization/year	$5,000.00	$5,000.00
maintenance/year	$500.00	$500.00
days/year	250	250
hours/day	8	8
labor/hour	$20.00	$20.00
molding cost/hour	$22.75	$22.75
No.parts/hour	240	30
cost/part	$0.09	$0.76

FINISHING COST

	Approach 1	Approach 2
cost/hour	$0.00	$50.00
parts/hour	240	30
cost/part	$0.00	$1.67

RESIN COST

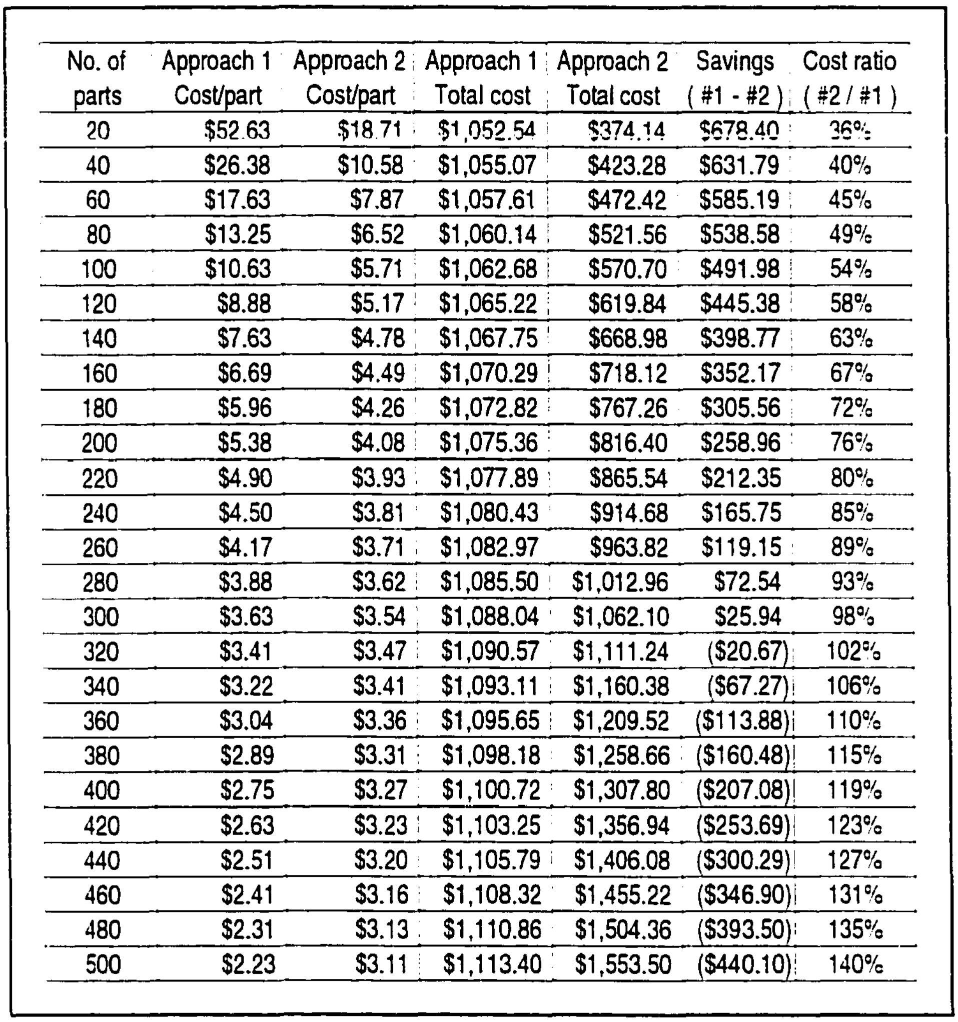
	Approach 1	Approach 2
cost/kg	$1.60	$1.60
kg/part	0.02	0.02
cost/part	$0.03	$0.03

No. parts = 50

Cost/part (#1) = $21.13
Cost/part (#2) = $8.96
Savings = $12.17
Cost ratio = 42%

Total cost (#1) = $1,056.34
Total cost (#2) = $447.85
Total saved = $608.49

Figure A.1 Computer spreadsheet for quick cost analysis.



No. of parts	Approach 1 Cost/part	Approach 2 Cost/part	Approach 1 Total cost	Approach 2 Total cost	Savings (#1 - #2)	Cost ratio (#2 / #1)
20	$52.63	$18.71	$1,052.54	$374.14	$678.40	36%
40	$26.38	$10.58	$1,055.07	$423.28	$631.79	40%
60	$17.63	$7.87	$1,057.61	$472.42	$585.19	45%
80	$13.25	$6.52	$1,060.14	$521.56	$538.58	49%
100	$10.63	$5.71	$1,062.68	$570.70	$491.98	54%
120	$8.88	$5.17	$1,065.22	$619.84	$445.38	58%
140	$7.63	$4.78	$1,067.75	$668.98	$398.77	63%
160	$6.69	$4.49	$1,070.29	$718.12	$352.17	67%
180	$5.96	$4.26	$1,072.82	$767.26	$305.56	72%
200	$5.38	$4.08	$1,075.36	$816.40	$258.96	76%
220	$4.90	$3.93	$1,077.89	$865.54	$212.35	80%
240	$4.50	$3.81	$1,080.43	$914.68	$165.75	85%
260	$4.17	$3.71	$1,082.97	$963.82	$119.15	89%
280	$3.88	$3.62	$1,085.50	$1,012.96	$72.54	93%
300	$3.63	$3.54	$1,088.04	$1,062.10	$25.94	98%
320	$3.41	$3.47	$1,090.57	$1,111.24	($20.67)	102%
340	$3.22	$3.41	$1,093.11	$1,160.38	($67.27)	106%
360	$3.04	$3.36	$1,095.65	$1,209.52	($113.88)	110%
380	$2.89	$3.31	$1,098.18	$1,258.66	($160.48)	115%
400	$2.75	$3.27	$1,100.72	$1,307.80	($207.08)	119%
420	$2.63	$3.23	$1,103.25	$1,356.94	($253.69)	123%
440	$2.51	$3.20	$1,105.79	$1,406.08	($300.29)	127%
460	$2.41	$3.16	$1,108.32	$1,455.22	($346.90)	131%
480	$2.31	$3.13	$1,110.86	$1,504.36	($393.50)	135%
500	$2.23	$3.11	$1,113.40	$1,553.50	($440.10)	140%

Figure A.2 Computer spreadsheet comparing costs of two different approaches.

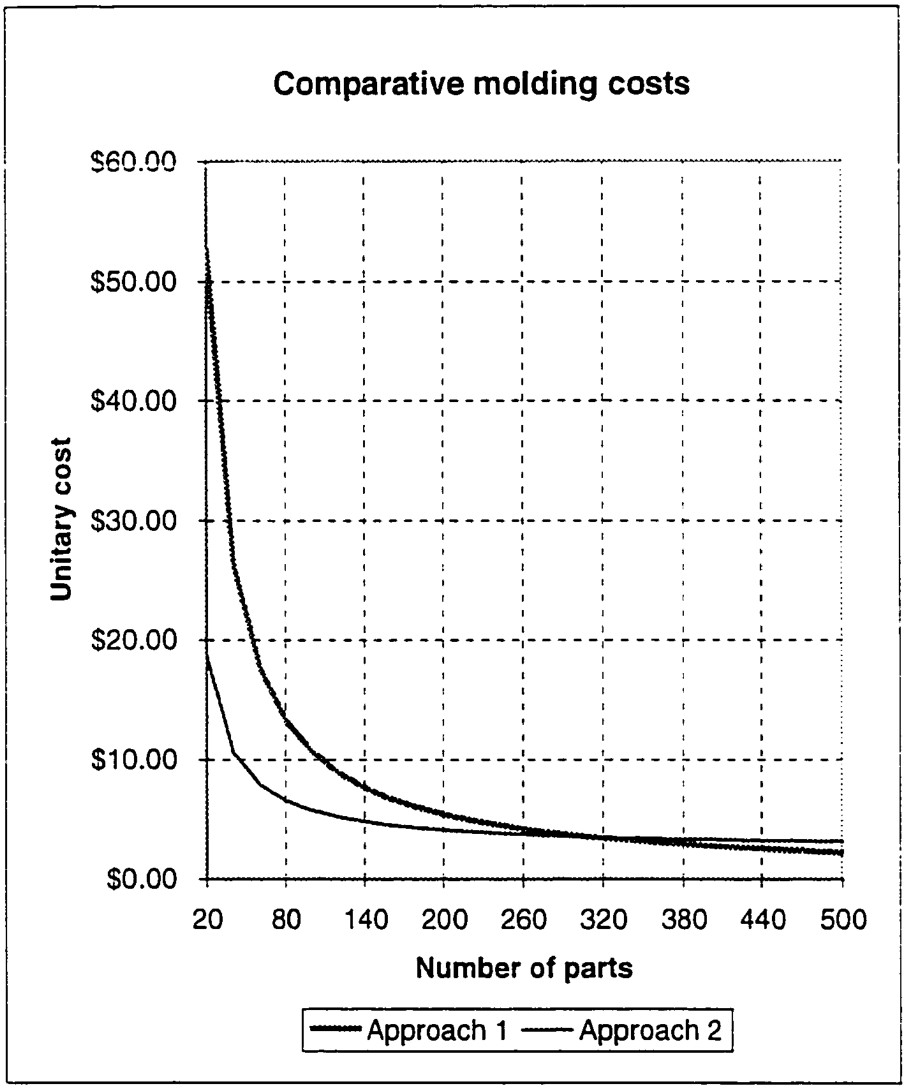


Figure A.3 Graph comparing costs of two different approaches.

APPENDIX B COOLING PLATE FOR THE SRP MOLD BASE

The cooling plate is made of two 1/4 inch aluminum plates. for maximum heat transfer. The groove and holes are machined on both plates. (see Figure B.1).

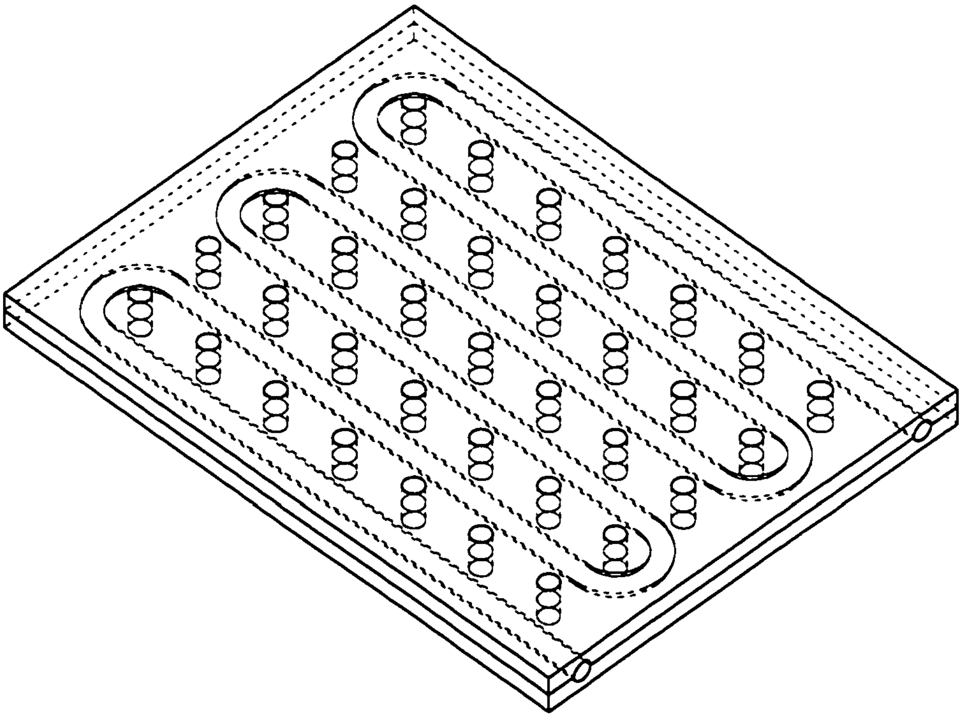


Figure B.1 A cooling plate for the mold base described in chapter 4.

Because of the symmetry of the design. the same CNC program is used to machine both halves of the cooling plate. The grid of holes is the same as in the mold base described in chapter 4. To avoid any leaks of water. silicone rubber can be applied along the cooling duct on one of the plates. The cooling plate is mounted to the mold base with bolts. They must be placed where they will not interfere with the dovetail clamps. or the mold itself.

APPENDIX C CNC CODE TO MACHINE THE MOLD

This is the CNC part program generated by "EZ-Mill", a module of the Computer Aided Manufacturing (CAM) package "EZ-CAM":

```
%:0111
(MKDXF4B 10-14-98)

(THE MAIN PROGRAM STARTS HERE)

(TOOL#2=0.1 DIA. AT Z-0.25" TOTAL DEPTH & -0." STEPS)
N50M6T2
N60G0G90G54X1.0Y-2.5S2500M3
N70G43Z0.1H2T1(GETTING THE NEXT TOOL READY)

(CENTER DRILLING OF THE HOLES)

N90Z0.1
N100G98G81Z-0.25R0.1F5.0
N110X1.5
N120X1.725Y-1.55
N130X1.5Y-0.5
N140X3.0Y-0.75
N150X2.667Y-1.638
N160X3.333
N170X4.5Y-2.5
N180X4.275Y-1.55
```

```
N190X4.5Y-0.5
N200X5.0
N210G80
N220G0G91G28Z0M19
N230M01
N240M6
(TOOL#1=0.25 DIA. AT Z-1.2" TOTAL DEPTH & -0.25" STEPS)
N260G0G90G54X1.0Y-2.5S2500M3
N270G43Z0.1H1T3(GETTING THE NEXT TOOL READY)

(DRILLING OF THE DEEP HOLES)

N290G98G83Z-1.2R0.1Q0.35F5.0
N300X1.5
N310X1.725Y-1.55
N320X1.5Y-0.5
N330X2.667Y-1.638
N340X3.333
N350X4.5Y-2.5
N360X4.275Y-1.55
N370X4.5Y-0.5
N380X5.0
N390G80
N400G0G91G28Z0M19
N410M01
N420M6
(TOOL#3=0.3125 DIA. AT Z-0.26" TOTAL DEPTH & -0." STEPS)
N440G0G90G54X3.0Y-0.75S2500M3
N450G43Z0.1H3T10(GETTING THE NEXT TOOL READY)

(DRILLING OF THE SPRUE WELL)
```

```
N470G98G81Z-0.26R0.1F5.0
N480G80
N490G0G91G28Z0M19
N500M01
N510M6
(TOOL#10=0.75 DIA. AT Z-0.02" TOTAL DEPTH & -0." STEPS)
N530G0G90G54X-0.5Y-2.9S1000M3
N540G43Z0.1H10T6(GETTING THE NEXT TOOL READY)

(MACHINING OF THE TOP SURFACE)

N560G1Z-0.02F5.0
N570X5.9F55.0
N580Y-0.1
N590X0.1
N600Y-2.6
N610X5.6
N620Y-0.4
N630X0.4
N640Y-2.3
N650X5.3
N660Y-0.7
N670X0.7
N680Y-2.0
N690X5.0
N700Y-1.0
N710X1.0
N720Y-1.7
N730X4.7
N740Y-1.3
N750X-0.5
N760G0G91G28Z0M19
```

```
N770M01
N780M6
(TOOL#6=0.25 DIA. AT Z-0.25" TOTAL DEPTH & -0.05" STEPS)
N800G0G90G54X1.675Y-1.625S3500M3
N810G43Z0.1H6

(MACHINING OF THE SLOT FOR THE SLIDING INSERTS)

N830G1Z-0.07F5.0
N840X4.325F55.0
N850Y-1.475
N860X1.675
N870Y-1.625
N880X1.575Y-1.725
N890X4.425
N900Y-1.375
N910X1.575
N920Y-1.725
N930X1.475Y-1.825
N940X4.525
N950Y-1.275
N960X1.475
N970Y-1.825
N980X1.375Y-1.925
N990X4.625
N1000Y-1.175
N1010X1.375
N1020Y-1.925
N1030G0
N1040X1.675Y-1.625
N1050G1Z-0.12F5.0
N1060X4.325F55.0
```

```
N1070Y-1.475
N1080X1.675
N1090Y-1.625
N1100X1.575Y-1.725
N1110X4.425
N1120Y-1.375
N1130X1.575
N1140Y-1.725
N1150X1.475Y-1.825
N1160X4.525
N1170Y-1.275
N1180X1.475
N1190Y-1.825
N1200X1.375Y-1.925
N1210X4.625
N1220Y-1.175
N1230X1.375
N1240Y-1.925
N1250G0
N1260X1.675Y-1.625
N1270G1Z-0.17F5.0
N1280X4.325F55.0
N1290Y-1.475
N1300X1.675
N1310Y-1.625
N1320X1.575Y-1.725
N1330X4.425
N1340Y-1.375
N1350X1.575
N1360Y-1.725
N1370X1.475Y-1.825
N1380X4.525
```

```
N1390Y-1.275
N1400X1.475
N1410Y-1.825
N1420X1.375Y-1.925
N1430X4.625
N1440Y-1.175
N1450X1.375
N1460Y-1.925
N1470G0
N1480X1.675Y-1.625
N1490G1Z-0.22F5.0
N1500X4.325F55.0
N1510Y-1.475
N1520X1.675
N1530Y-1.625
N1540X1.575Y-1.725
N1550X4.425
N1560Y-1.375
N1570X1.575
N1580Y-1.725
N1590X1.475Y-1.825
N1600X4.525
N1610Y-1.275
N1620X1.475
N1630Y-1.825
N1640X1.375Y-1.925
N1650X4.625
N1660Y-1.175
N1670X1.375
N1680Y-1.925
N1690G0
N1700X1.675Y-1.625
```

```
N1710G1Z-0.27F5.0
N1720X4.325F55.0
N1730Y-1.475
N1740X1.675
N1750Y-1.625
N1760X1.575Y-1.725
N1770X4.425
N1780Y-1.375
N1790X1.575
N1800Y-1.725
N1810X1.475Y-1.825
N1820X4.525
N1830Y-1.275
N1840X1.475
N1850Y-1.825
N1860X1.375Y-1.925
N1870X4.625
N1880Y-1.175
N1890X1.375
N1900Y-1.925
N1910G0Z-0.17
N1920X2.55Y-1.625

(MACHINING OF THE POCKET)

N1940G1Z-0.32F5.0
N1950X3.45F55.0
N1960Y-1.4875
N1970G3X3.4375Y-1.475I-0.0125J0.
N1980G1X2.5625
N1990G3X2.55Y-1.4875I0.J-0.0125
N2000G1Y-1.625
```

```
N2010X2.45Y-1.725
N2020X3.55
N2030Y-1.4875
N2040G3X3.4375Y-1.375I-0.1125J0.
N2050G1X2.5625
N2060G3X2.45Y-1.4875I0.J-0.1125
N2070G1Y-1.725
N2080X2.35Y-1.825
N2090X3.65
N2100Y-1.4875
N2110G3X3.4375Y-1.275I-0.2125J0.
N2120G1X2.5625
N2130G3X2.35Y-1.4875I0.J-0.2125
N2140G1Y-1.825
N2150X2.25Y-1.925
N2160X3.75
N2170Y-1.4875
N2180G3X3.4375Y-1.175I-0.3125J0.
N2190G1X2.5625
N2200G3X2.25Y-1.4875I0.J-0.3125
N2210G1Y-1.925
N2220G0
N2230X2.55Y-1.625
N2240G1Z-0.37F5.0
N2250X3.45F55.0
N2260Y-1.4875
N2270G3X3.4375Y-1.475I-0.0125J0.
N2280G1X2.5625
N2290G3X2.55Y-1.4875I0.J-0.0125
N2300G1Y-1.625
N2310X2.45Y-1.725
N2320X3.55
```

```
N2330Y-1.4875
N2340G3X3.4375Y-1.375I-0.1125J0.
N2350G1X2.5625
N2360G3X2.45Y-1.4875I0.J-0.1125
N2370G1Y-1.725
N2380X2.35Y-1.825
N2390X3.65
N2400Y-1.4875
N2410G3X3.4375Y-1.275I-0.2125J0.
N2420G1X2.5625
N2430G3X2.35Y-1.4875I0.J-0.2125
N2440G1Y-1.825
N2450X2.25Y-1.925
N2460X3.75
N2470Y-1.4875
N2480G3X3.4375Y-1.175I-0.3125J0.
N2490G1X2.5625
N2500G3X2.25Y-1.4875I0.J-0.3125
N2510G1Y-1.925
N2520G0
N2530X2.55Y-1.625
N2540G1Z-0.39F5.0
N2550X3.45F55.0
N2560Y-1.4875
N2570G3X3.4375Y-1.475I-0.0125J0.
N2580G1X2.5625
N2590G3X2.55Y-1.4875I0.J-0.0125
N2600G1Y-1.625
N2610X2.45Y-1.725
N2620X3.55
N2630Y-1.4875
N2640G3X3.4375Y-1.375I-0.1125J0.
```

```
N2650G1X2.5625

N2660G3X2.45Y-1.4875I0.J-0.1125

N2670G1Y-1.725

N2680X2.35Y-1.825

N2690X3.65

N2700Y-1.4875

N2710G3X3.4375Y-1.275I-0.2125J0.

N2720G1X2.5625

N2730G3X2.35Y-1.4875I0.J-0.2125

N2740G1Y-1.825

N2750X2.25Y-1.925

N2760X3.75

N2770Y-1.4875

N2780G3X3.4375Y-1.175I-0.3125J0.

N2790G1X2.5625

N2800G3X2.25Y-1.4875I0.J-0.3125

N2810G1Y-1.925

N2820G0Z0.08

N2830X3.0Y-1.3


(MACHINING OF THE GATE)


N2850G1Z-0.04F5.0

N2860Y-0.75F55.0

N2870G0G91G28Z0M19

N2880G28X0Y0

N2890M01

N2900M6(PUTTING THE FIRST TOOL BACK INTO SPINDLE)

N2910M30

%
```

APPENDIX D THEORETICAL ANALYSIS OF THERMAL CONDUCTIVITY IN HYBRID MACHINED/CAST MOLDS

Thermal conductivity behavior of aluminum-filled epoxy

Numerous theoretical and empirical correlations are found in the literature for predicting the thermal conductivity of particulate-filled polymers. A careful review of these models by Progelhof, Throne and Ruetsch indicates that no one correlation or technique predicts accurately the thermal conduction of all types of composites. Their investigation found however that for a solid-filled composite (such as aluminum-filled epoxy), the Lewis and Nielsen equation fits the experimental data better, for the range of fillers they tested [52].

The models that shown here are:

- The Series Model,
- the Parallel Model,
- the Geometric Mean Model, and
- the Lewis and Nielsen Semi-Theoretical Model.

Series Model:

$$Kc = (1 - \phi)Km + \phi Kf$$

Parallel Model:

$$\frac{1}{Kc} = \frac{(1 - \phi)}{Km} + \frac{\phi}{Kf}$$

Geometric Mean Model:

$$Kc = Kf^{\phi} Km^{(1-\phi)}$$

Lewis and Nielsen Semi-Theoretical Model [53]:

$$Kc = Km\frac{1 + AB\phi}{1 - B\phi\psi}$$

where:

$$A = K_E - 1$$

$$B = \frac{\frac{Kf}{Km} - 1}{\frac{Kf}{Km} + A}$$

$$\psi = 1 + \frac{1 - \phi_f}{\phi_f^2}\phi$$

In these equations, Kc, Km and Kf are the thermal conductivities of the composite, the matrix, and filler, respectively. ϕ is the volume fraction of the filler. A is a constant related to the generalized Einstein coefficient K_E. B is a constant related to the relative conductivity of the components. ψ is a function related to the maximum packing fraction ϕ_f of the filler. ϕ_f is the ratio of the density of the filler material to the maximum density of the bulk powder. The values for A and ϕ_f for many geometric shapes and orientation are given in tables in the article by Progelhof, Throne and Ruetsch. In this particular case the value for A was chosen to be 1.5, assuming that the aluminum powder were spheres. The value for ϕ_m was estimated to be 0.56.

The graphs of the relative thermal conductivity prediction of the Parallel model, the Geometric Mean model, and the Lewis and Nielsen model are shown in Figure D.1. The Series model is not shown because it is quite inaccurate for two materials with extremely different thermal conductivities. It is basically just a straight line.

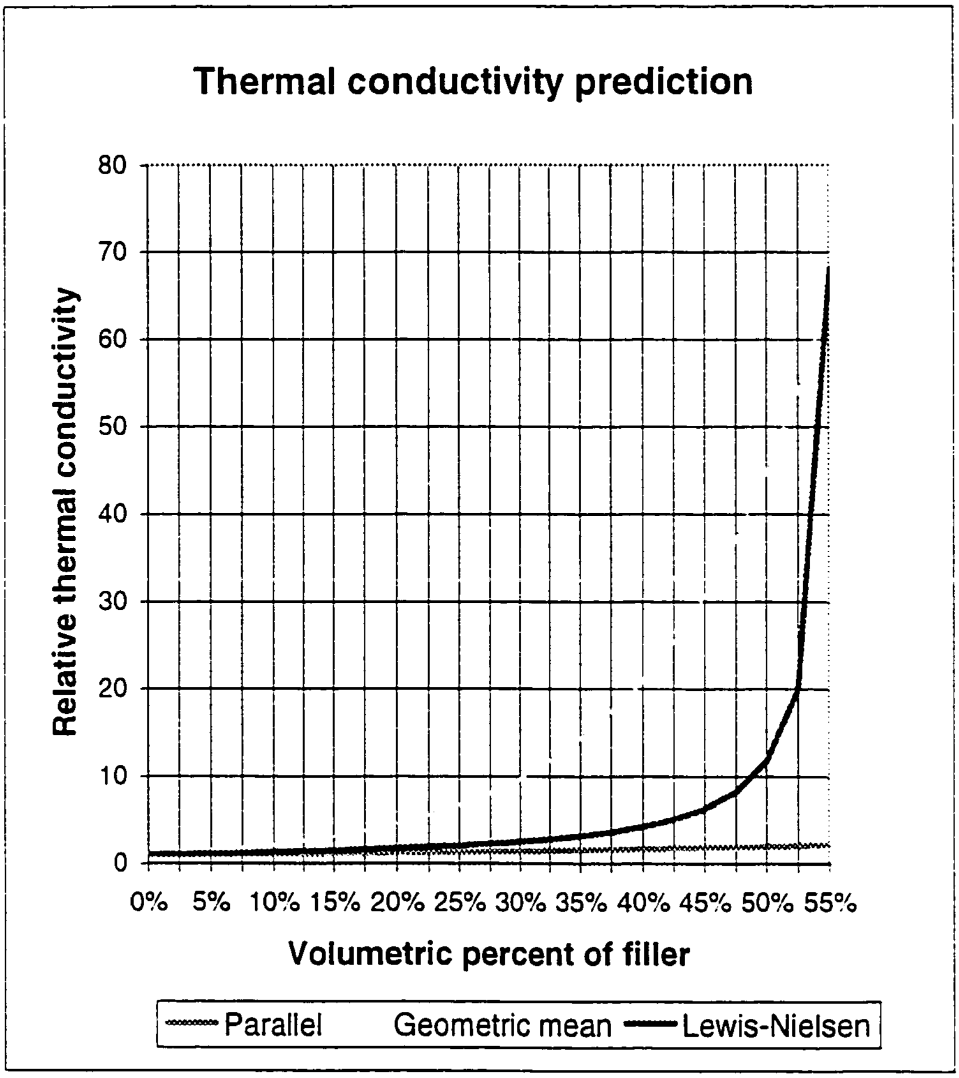


Figure D.1 Thermal conductivity prediction for a composite.

BIBLIOGRAPHY

[1] T. T. Wohlers. "New report finds 34% unit sales growth in rapid prototyping industry." February 1998. Press release found at http://lamar.colostate.edu/~wohlers/.

[2] T. T. Wohlers. "Rapid prototyping state of the industry. 1997 worldwide progress report. Executive summary.," in *Proceedings of the Rapid Prototyping and Manufacturing'97 Conference*, (Dearborn, Michigan), RPA-SME, Society of Manufacturing Engineers, April 1997.

[3] P. Blake. "Accelerating product development using rapid prototyping systems." *Rapid News - Time-Compression Technologies*, vol. 2, pp. 38–41, September 1997.

[4] P. J. C. Gough. "From A to B with very little 't'." *Rapid News - Time-Compression Technologies*, vol. 1, pp. 16–21, December 1996.

[5] K. Ishii, *Mechanical Design: Theory and Methodology*, ch. Life-Cycle Design, pp. 312–328. New York: Springer–Verlag, 1996.

[6] M. Burns, *Automated Fabrication*. Englewood Cliffs, New Jersey: Prentice–Hall, 1993.

[7] D. Kochan, ed., *Solid Freeform Manufacturing. Advanced Rapid Prototyping.*, vol. 19 of *Manufacturing Research and Technology*. New York: Elsevier, 1993.

[8] J. J. Beaman, J. W. Barlow, D. L. Bourell, R. H. Crawford, H. L. Marcus, and K. P. McAlea, *Solid Freeform Fabrication*. Boston, Massachusetts: Kluwer Academic Publishers, 1997.

[9] M. Burns. "The origins and direction of the Fabricator revolution." *Rapid News - Time-Compression Technologies*, vol. 2, pp. 32–7, September 1997.

[10] P. Gupta and J. L. Hall, *Rapid Prototyping Systems - fast track to product realization*, ch. Rapid Prototyping in Die Manufacturing, pp. 3–17. Dearborn, Michigan: Society of Manufacturing Engineers, 1994.

[11] P. F. Jacobs, "Recent advances in rapid tooling from stereolithography." 3D Systems white paper, Valencia, CA, October 1996.

[12] Albright Technologies, Inc., "Rapid tooling technology." 1997. Information found at http://albright1.com/.

[13] EOS, GmbH, "EOSINT M builds cores and cavities for injection moulding in record time.." March 1997. Information found at: http://www.eos-gmbh.de/.

[14] CEMCOM Corp., "NCC rapid tooling for injection and compression molding." Information found at: http://www.cemcom.com/.

[15] S. Ashley, "RP industry's growing pains," *Mechanical Engineering*, vol. 120, pp. 64–67, July 1998.

[16] J. Mazumder, J. Choi, K. Nagarathnam, J. Koch, and D. Hetzner, "The direct metal deposition of H13 tool steel for 3-D components," *JOM: the Journal of the Minerals, Metals and Materials Society*, vol. 49, pp. 55–60, May 1997.

[17] D. Keicher and J. E. Smugeresky, "The laser forming of metallic components using particulate materials," *JOM: the Journal of the Minerals, Metals and Materials Society*, vol. 49, pp. 51–54, May 1997.

[18] G. Hatzilias, "Actua MJM Rapid Tooling," Georgia Tech's ME 8501 Special Problems report, November 1996. Information found at http://rpmi.marc.gatech.edu/~rapid_tooling/art.html.

[19] K. Filipiak and M. Kotnis, "Injection molding thermoplastic parts in days," *Rapid News - Time-Compression Technologies*, vol. 3, pp. 46–53, September 1998.

[20] R. Merz, F. Prinz, K. Ramaswami, M. Terk, and L. Weiss, "Shape deposition manufacturing," in *Proceedings of the 1994 Solid Freeform Fabrication Symposium*, (Austin, Texas), University of Texas at Austin, August 1994.

[21] K. Hartmann, R. Krishnan, R. Merz, G. Neplotnik, F. Prinz, L. Schultz, M. Terk, and L. Weiss, "Robot-assited shape deposition manufacturing," in *Proceedings of the 1994 IEEE International Conference on Robotics and Automation*, (San Diego, California), May 1994.

[22] T. Moran, "Routing out the details saves time and money," *American Machinist*, vol. 141, pp. 47+, July 1997.

[23] T. Moran, "Alternative rapid prototyping - combining stereolithography with CNC routers for higher precision," *Rapid News - Time-Compression Technologies*, vol. 2, pp. 41-5, July/August 1997.

[24] E. A. Judson and T. L. Starr, "Rapid prototyping with ceramics using injection molding and stereolithography," Georgia Institute of Technology, http://rpmi.marc.gatech.edu/.

[25] J. Metelnick, "Don't forget CNC in rapid prototyping," *Machine Design*, vol. 66, pp. 98-101, November 7 1994.

[26] J. W. Schmidt, "CNC machining keeps pace with new technologies," *Automotive Engineering*, vol. 105, pp. 97-99, July 1997.

[27] Y.-A. Son, S. Park, and S. Ha, "Rapid tooling vs. five-axis milling for ship propeller prototypes," *Rapid Prototyping*, vol. 4, no. 3, pp. 1-4, 1998.

[28] Morgan Industries, Inc., *Cutting Costs in Short-Run Plastics Injection Molding*. El Segundo, California: T/C Publications, 1985.

[29] R. A. Malloy, *Plastic Part Design for Injection Molding*. Cincinnati, Ohio: Hanser/Gardner Publications, 1994.

[30] P. Schröer. *Mold-Making Handbook*. ch. Injection Molds. p. 50. New York: Hanser Publishers. 1983.

[31] J. B. Dym. *Injection Molds and Molding: A Practical Manual*. New York: Van Nostrand Reinhold. second ed.. 1987.

[32] W. Hartmann. *Injection Moulds*. ch. Mould Venting. pp. 85–93. Düsseldorf: VDI–Verlag GmbH. 1980.

[33] H. Gastrow. *Injection Molds: 102 Proven Designs*. New York: Hanser Publishers. 1983.

[34] J. Klees. *Fundamentals of mold design. gating and cooling*. Society of Plastics Engineers. Seminar presented in Des Moines. Iowa. June 1996.

[35] K. Kurz. K. Craig. and B. Wolf. "Developing a flexible automated fixturing device." *Mechanical Engineering*. vol. 116. pp. 59–63. July 1994.

[36] S. K. Hargrove. "Characteristics of job shop operations in fixture design." *Industrial Engineering*. vol. 26. p. 26. December 1994.

[37] R. J. Menassa and W. R. DeVries. "Optimization methods applied to selecting support positions in fixture design." *Journal of Engineering for Industry*. vol. 113. pp. 412–18. November 1991.

[38] Y. K. Rong and Y. Bai. "Automated generation of fixture configuration design." *Journal of Manufacturing Science and Engineering*. vol. 119. pp. 208–19. May 1997.

[39] A. J. Trappey and S. Matrubhutam. "Fixture configuration using projective geometry." *Journal of Manufacturing Systems*. vol. 12. no. 6. pp. 486–95. 1993.

[40] B. Shirinzadeh. "Strategies for planning and implementation of flexible fixturing systems in a computer integrated manufacturing environment." *Computers in Industry*. vol. 30. pp. 175–83. October 15 1996.

[41] T. Hanada, B. Bandyopadhyay, and T. Hoshi, "Implementation of low-volume FMS for prismatic components," *Journal of Manufacturing Systems*, vol. 14, no. 2, pp. 91–108, 1995.

[42] R. G. Martin, "Quick change system for mold bases," U.S. Patent 5,350,289, September 1994.

[43] R. E. Pleasant, *Round Mate Design Manual*. Pleasant Precision, Inc., Kenton, Ohio, 1993. Technical manual.

[44] J. De Gaspari, "'Modular molding' saves on small parts," *Plastics Technology*, vol. 39, p. 84, December 1993.

[45] E. G. Hoffman, *Modular Fixturing*. Lake Geneva, Wisconsin: Manufacturing Technology Press, 1987.

[46] W. E. Boyes, ed., *Low-cost Jigs, Fixtures and Gages for Limited Production*. Dearborn, Michigan: Society of Manufacturing Engineers, 1986.

[47] T. J. Drozda, ed., *Jigs and Fixtures*. Dearborn, Michigan: Society of Manufacturing Engineers, third ed., 1989.

[48] J. Abou-Hanna, K. Okamura, and T. McGreevy, "Sinkage characteristics of workpieces in flexible particulate bed fixtures: an experimental and numerical investigation," *Journal of Manufacturing Systems*, vol. 13, no. 5, pp. 359–69, 1994.

[49] J. Abou-Hanna, K. Okamura, and T. McGreevy, "Experimental study of static and dynamic rigidities of flexible particulate bed fixtures under external vertical and torque loads," *Journal of Manufacturing Systems*, vol. 13, no. 3, pp. 177–89, 1994.

[50] C. A. Harper, ed., *Handbook of Plastics, Elastomers and Composites*. New York: McGraw-Hill, third ed., 1996.

[51] R. Hope, R. Roth, and P. Jacobs, "Adaptive slicing with sloping layer surfaces," *Rapid Prototyping Journal*, vol. 3, no. 3, pp. 89–98, 1997.

[52] R. C. Progelhof, J. L. Throne, and R. R. Ruetsch, "Methods for predicting the thermal conductivity of composite systems: A review," *Polymer Engineering and Science*, vol. 16, pp. 615–25, September 1976.

[53] L. Nielsen, "Thermal conductivity of particulate-filled polymers," *Journal of Applied Polymer Science*, vol. 17, pp. 3819–20, 1973.