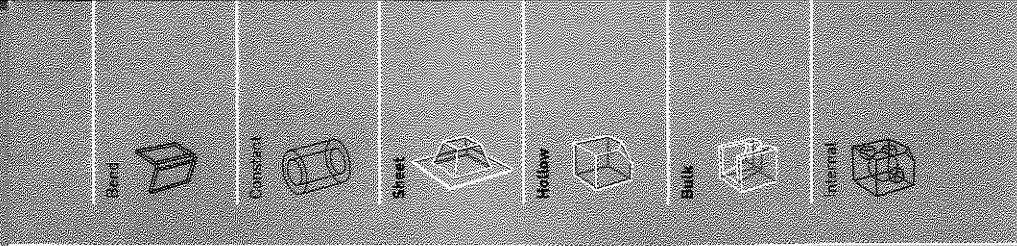
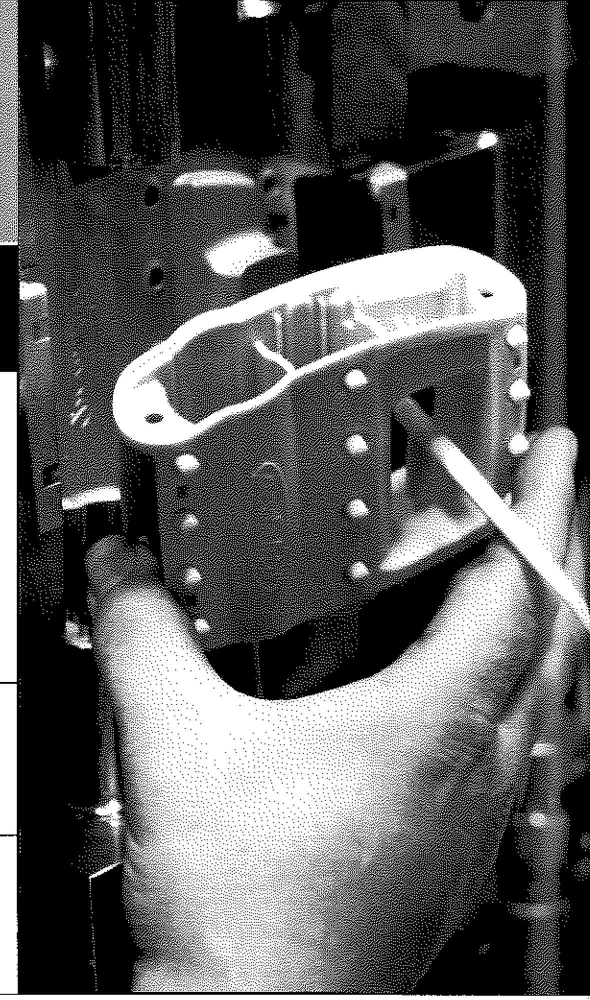





# Injection Molding



**This is one of the leading processes used for manufacturing plastic products, and is ideal for high volume production of identical products. Variations on conventional injection molding include gas assisted, multishot and in-mold decoration.**

| Costs | Typical Applications | Suitability |
|---|---|---|
| • Very high tooling costs but depends on complexity and number of cavities<br>• Very low unit costs | • Automotive<br>• Consumer electronics and appliances<br>• Industrial and household products | • High volume mass production |
| **Quality** | **Related Processes** | **Speed** |
| • Very high surface finish<br>• Highly repeatable process | • Reaction injection molding<br>• Thermoforming<br>• Vacuum casting | • Injection cycle time is generally between 30 and 60 seconds |




## INTRODUCTION

Injection molding is a widely used and well-developed process that is excellent for rapid production of identical parts with tight tolerances. It is used to create a huge diversity of our day-to-day plastic products. Accurately engineered tools and high injection pressures are essential for achieving excellent surface finish and reproduction of detail. Consequently, this process is suitable only for high volume production runs.

There are many different variations on injection molding technology. Some of the most popular include gas-assisted injection molding (page 58), multishot injection molding (page 60) and in-mold decoration (page 62).

## TYPICAL APPLICATIONS

Injection-molded parts can be found in every market sector, in particular in automotive, industrial and household products. They include shopping baskets, stationery, garden furniture, keypads, the housing of consumer electronics, plastic cookware handles and buttons.

## RELATED PROCESSES

The relative suitability of related processes depends on factors such as part size and configuration, materials being used, functional and aesthetic requirements, and budget.

Although injection molding is very often the most desirable process due to its repeatability and speed, thermoforming (page 30) is a suitable alternative for certain sheet geometries, and extrusion is more cost-effective for the production of continuous profiles.

Parts that will ultimately be made by injection molding can be prototyped and produced in low volumes by vacuum casting (page 40) and reaction injection molding (page 64). Both of these processes are used to form polyurethane resin (PUR). This is a thermosetting plastic that is available in a wide range of grades, colours and hardnesses. It can be solid or foamed. Reaction injection molding is used for a diverse range of products, including foam moldings for upholstering furniture and car seats, and low volume production of car bumpers and dashboards.

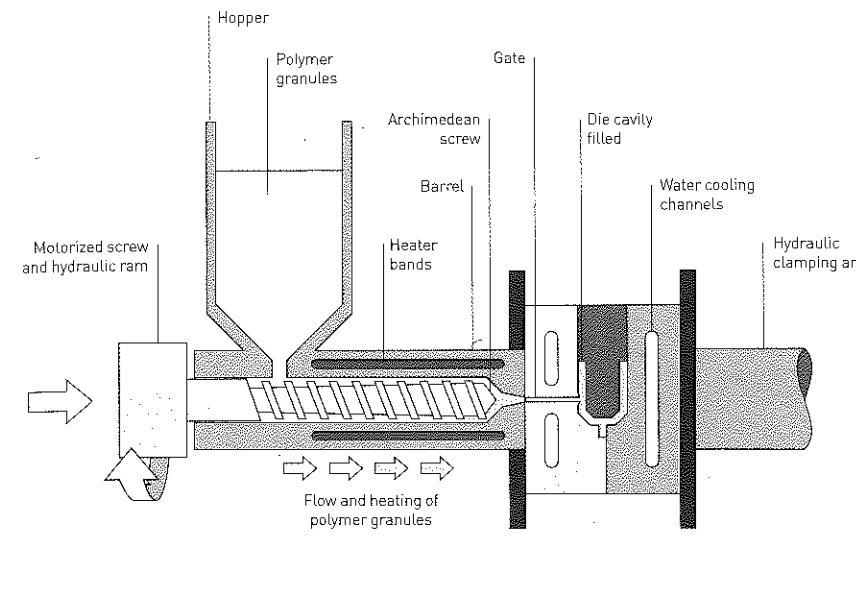
## QUALITY

The high pressures used during injection molding ensure good surface finish, fine reproduction of detail and, most importantly, excellent repeatability.

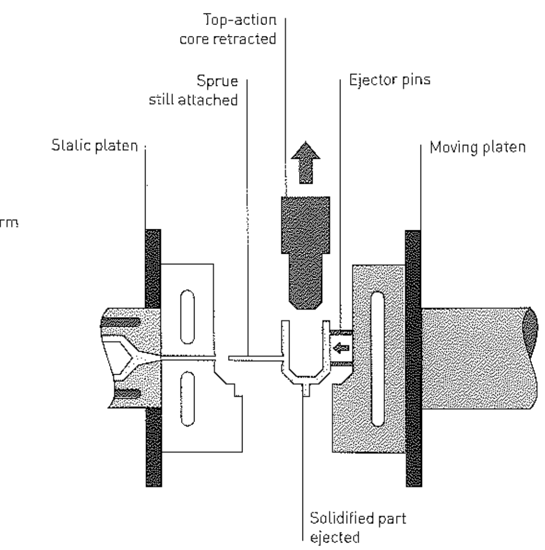
The downside of the high pressure is that the resolidified polymer has a tendency to shrink and warp. These defects can be designed out using rib details and careful flow analysis.

Surface defects can include sink marks, weld lines and streaks of pigment. Sink marks occur on the surface opposite a rib detail, and weld lines appear where the material is forced to flow around obstacles, such as holes and recesses.

## Injection Molding Process




Injection molding process




Ejection cycle

## DESIGN OPPORTUNITIES

So much is practically possible with injection molding that restrictions generally come down to economics. The process is least expensive when using a simple split mold. Most expensive are very complex shapes, which are achievable in a range of sizes, from large car bumpers to the tiniest widgets. Retractable cores controlled by cams or hydraulics can make undercuts from the sides, top or bottom of the tool simultaneously and will not affect the cost significantly, depending on the complexity of the action.

In-mold and insert film decoration are often integrated into the molding cycle, so eliminating finishing processes such as printing. There is also a range of pigments available to produce metallic, pearlescent, thermochromatic and photochromatic effects, as well as vibrant fluorescent and regular colour ranges. Inserts and snap-fits can be molded into the product to assist assembly.

Multishot injection molding can combine up to 6 materials in one product. The combination possibilities include density, rigidity, colour, texture and varying levels of transparency.

**TECHNICAL DESCRIPTION**

**Polymer granules are dried to exactly the right water content and fed into the hopper. Any pigments are added at this stage at between 0.5% and 5% dilution.**

**The material is fed into the barrel, where it is simultaneously heated, mixed and moved towards the mold by the rotating action of the Archimedean screw. The melted polymer is held in the barrel momentarily as the pressure builds up ready for injection into the mold cavity.**

**The correct pressure is achieved and the melted plastic is injected into the die cavity. Cycle time is determined by the size of the part and how long the polymer takes to resolidify, and is usually between 30 and 60 seconds.**

**Clamping pressure is maintained after injection to minimize warpage and shrinkage once the part is ejected.**

**To eject the part, the tools move apart, the cores retract and force is applied by the ejector pins to separate the part from the surface of the tool. The part is dispensed onto a conveyor belt or holding container, sometimes by a robotic arm.**

**Tools and cores are generally machined from either aluminium or tool steel. The tools are very complex parts of the injection molding process. They are made up of water cooling channels (for temperature control), an injection point (gate), runner systems (connecting parts) and electronic measuring equipment which continuously monitors temperature. Good heat dispersal within the tool is essential to ensure the steady flow of melted polymer through the die cavity. To this end, some cores are machined from copper, which has much better conductive qualities than aluminium or steel.**

**The least expensive injection molding tooling consists of 2 halves, known as the male tool and female tool. But engineers and toolmakers are constantly pushing the boundaries of the process with more complex tooling, retractable cores, multiple gates and multishot injection of contrasting materials.**

### DESIGN CONSIDERATIONS

Designing for injection molding is a complex and demanding task that involves designers, polymer specialists, engineers, toolmakers and molders. Full collaboration by these experts will help realize the many benefits of this process.

Injection molding operates at high temperatures and injects plasticized material into the die cavity at high pressure. This means that problems can occur as a result of shrinkage and stress build-up. Shrinkage can result in warpage, distortion, cracking and sink marks. Stress can build up in areas with sharp corners and draft angles that are too small. Draft angles should be at least 0.5° to avoid stressing the part during ejection from the tool.

The injected plastic will follow the path of least resistance as it enters the die cavity and so the material must be fed into the thickest wall section and finish in the areas with the thinnest wall sections. For best results wall thickness should be uniform, or at least within 10%. Uneven wall sections will produce different rates of cooling, which cause the part to warp. The factors that determine optimal wall thickness include cost, functional requirements and molding consideration.

Ribs serve a dual function in part design: firstly, they increase the strength of the part, while decreasing wall thickness; and secondly, they aid the flow of material during molding. Ribs should not exceed 5 times the height of the wall thickness. Therefore, it is often recommended to use lots of shallow ribs, as opposed to fewer deep ribs.

All protruding features are treated as ribs and must be 'tied-in' (connected) to the walls of the part to reduce air traps and possible stress concentration points. Holes and recesses are often integrated in part design in order to avoid costly secondary operations.

Injection molded parts are often finished with a fine texture, which disguises surface imperfections. Large gloss areas are more expensive to produce than matt or textured ones.

### COMPATIBLE MATERIALS

Almost all thermoplastic materials can be injection molded. It is also possible to mold certain thermosetting plastics and metal powders in a polymer matrix.

### COSTS

Tooling costs are very high and depend on the number of cavities and cores and the complexity of design.

Injection molding can produce small parts very rapidly, especially because multicavity tools can be used to increase production rates dramatically. Cycle time is between 30 and 60 seconds, even for multiple cavity tools. Larger parts have longer cycle times, especially because the polymer will take longer to resolidify and so will need to be held in the tool while it cools.

Labour costs are relatively low. However, manual operations, such as mold preparation and demolding, increase the costs significantly.

### ENVIRONMENTAL IMPACTS

Thermoplastic scrap can be directly recycled in this process. Some applications, such as medical and food packaging, require a high level of virgin material, whereas garden furniture may require only 50% virgin material for adequate structural integrity, hygiene and colouring capability.

Injection molded plastic is commonly associated with mass produced short term products, such as disposables. However, it is possible to design products so that they can be disassembled easily, which is advantageous for both maintenance and recycling. If different types of materials are used, then snap fits and other mechanical fasteners make it more convenient to disassemble and dispose of the parts with minimal environmental impact.

Case Study

## Manufacturing and assembling a Pedalite

This bicycle pedal light is powered by an energy storage capacitor and microgenerator rather than any form of chemical battery. It was designed by Product Partners, in conjunction with the client (Pedalite Limited), toolmaker/molder (ENL Limited), gearbox manufacturer (Davall Gears) and polymer distributor (Distrupol Limited).

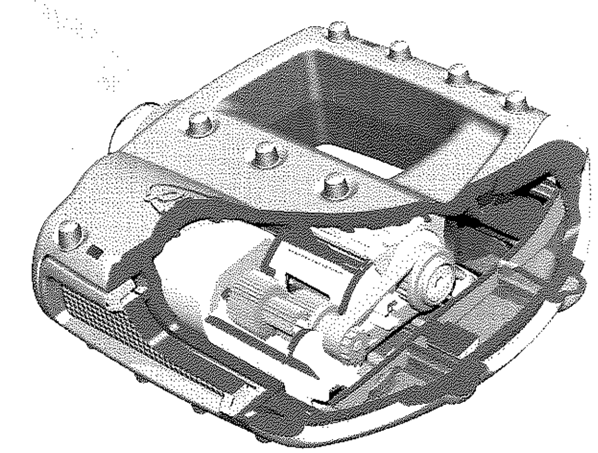
A close working partnership with ENL ensured Product Partners' ideas were successfully translated through the design, development and toolmaking process, ensuring a 'right first time' product. The technically challenging combined overmold assembly, for instance, was approved at the first-off tool trials.

To ensure the gearbox concept was feasible Davall Gears were recruited to provide expertise in gear ratios, gear detail design, materials specification and manufacture. Polyamide (PA) nylon was chosen for its exceptional wear characteristics and self-lubrication properties.

The cutaway drawing (below) shows the anatomy of the injection-molded parts as well as the internal mechanisms and parts. Injection-molded casings very often have to accommodate a fixed-space package. In this case, specific spindle bearings have been used to satisfy legislations and the gear system is designed for optimum energy generation.

**MOLDFLOW ANALYSIS**

Polymer distributor Distrupol was consulted about materials selection and moldflow analysis. Feedback on the developing design led to the modification of the components in CAD to reduce potential problems of sinkage, flow marks, weld lines and so forth.

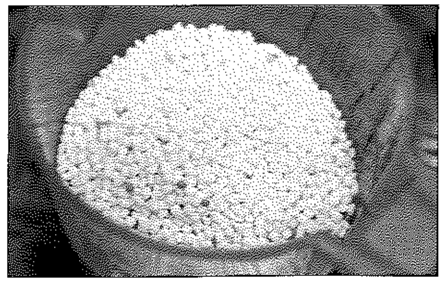


**MOLDING THE PEDALITE**

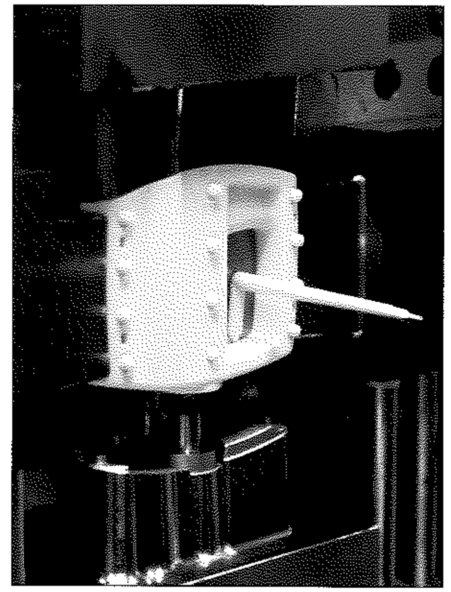
The raw material is a glass-filled nylon that is white in its raw state. If a colour is required, then pigment is added. In this case a small quantity of Clariant masterbatch yellow was used (**image 1**). The end result is a surprisingly vivid yellow colouring.

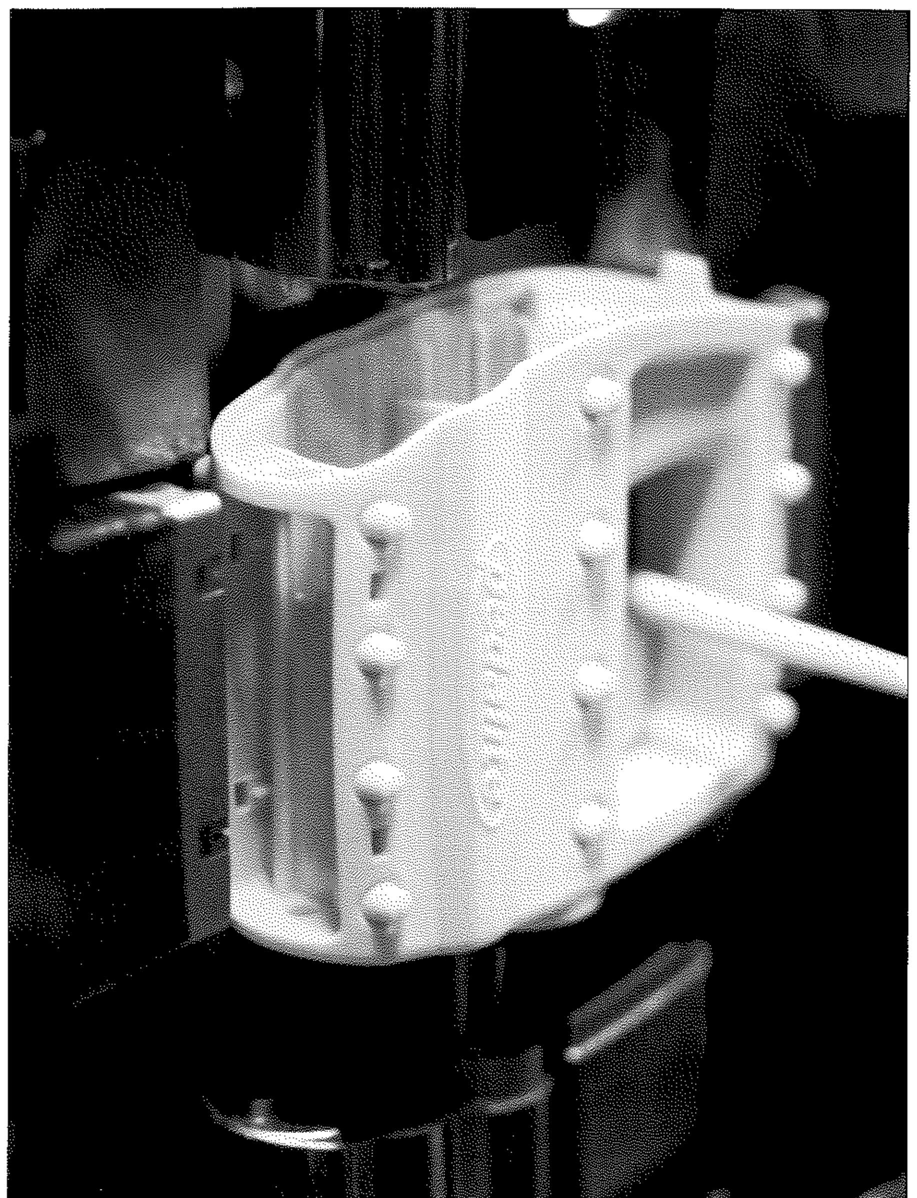
In normal operations, the injection molding process takes place behind a screen within the machine (**image 2**), but for the purposes of this case study the dies are shown in close-up with the screen open.

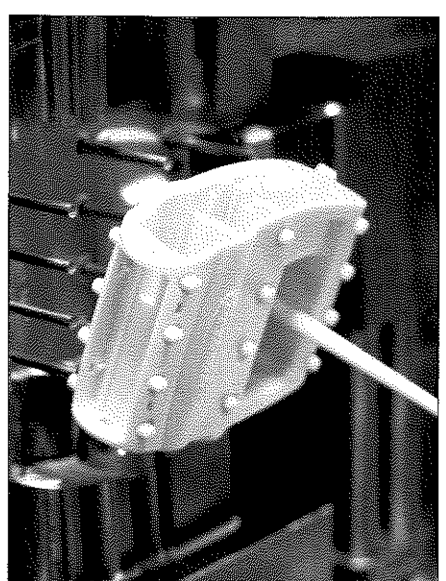
The polymer is melted and mixed before injection into the die cavity. Once the die cavity has been filled, packed and clamped, and the polymer has resolidified, the male and female halves of the mold move apart. The product is held in the moving tool by the upper and lower retractable cores and the 2 side-action cores (**image 3**). The injection point is indicated by the sprue, which has remained intact, to be removed either by hand or robotically. The 4 cores are retracted in sequence, to reveal the true complexity of this molding (**image 4**). Finally, the part is ejected from the mold by a series of ejector pins (**image 5**).


1


2


3


4

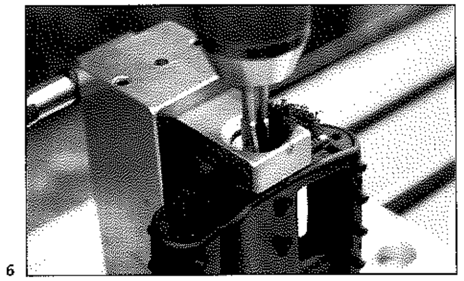

5

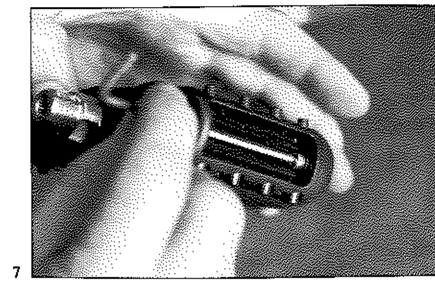
**ASSEMBLING THE PEDALITE**

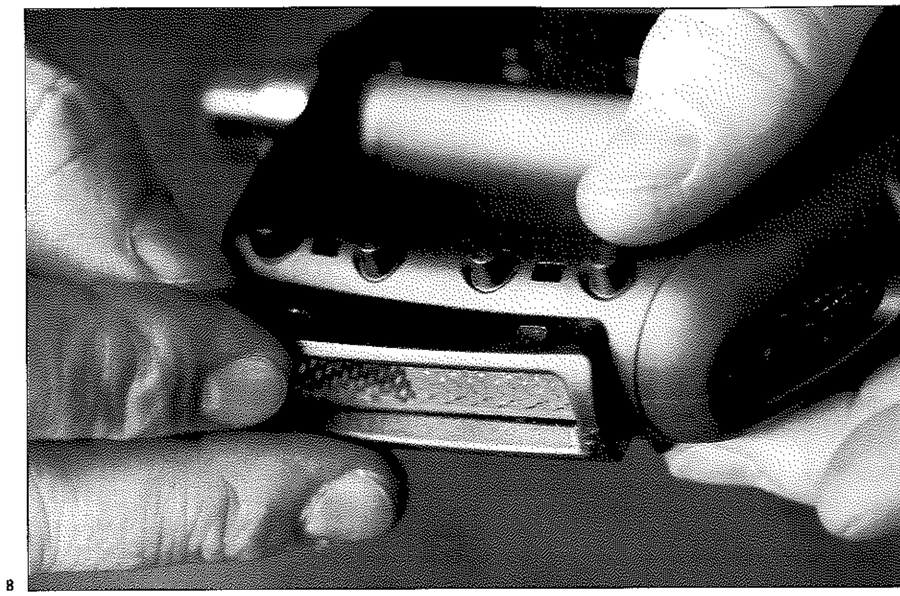
There are many parts that make up the Pedalite (see image, page 54). All of the plastic parts are injection molded. The bearing locator is a friction fit, which requires more precise tolerance than can be achieved with injection molding. Therefore, it is drilled post-forming (**image 6**) and the bearing locator and bearing inserted. The overmolded end cap is fixed to the pedal housing with screw fixings (**image 7**). The reflectors snap fit into place (**image 8**), ensuring that all the components are held together securely. The snap fits can be released so that the Pedalite can be dismantled for maintenance and recycling (**image 9**). The finished product is installed by conventional means onto a bicycle (**image 10**).

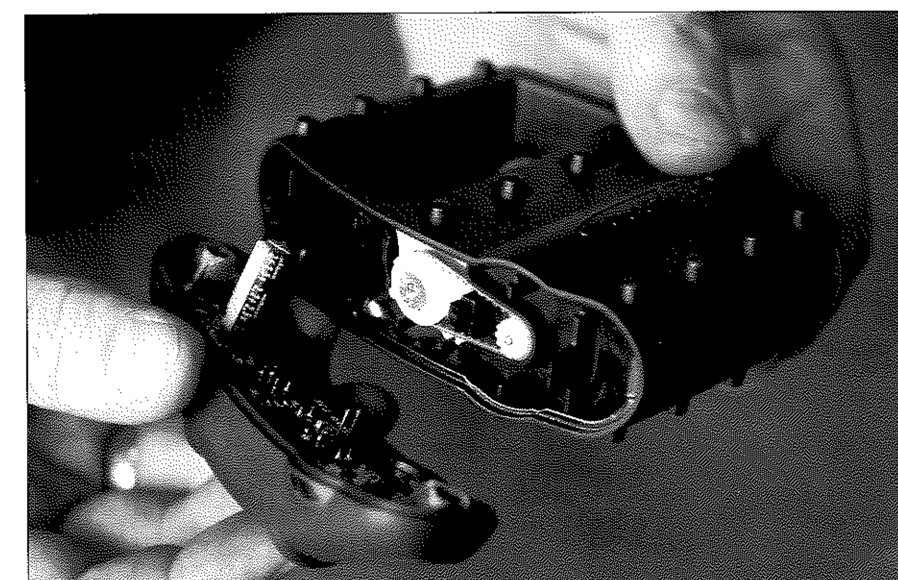
Cycle pedal design is subject to considerable safety and technical restrictions. Pedalite eliminates the expense and inconvenience of battery replacement, as well as the negative environmental impact of battery disposal.

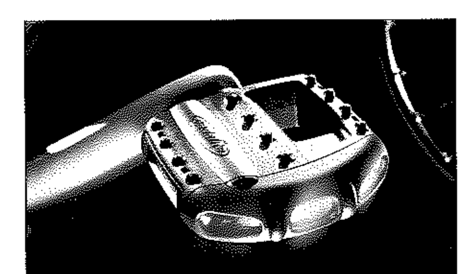
The 24/7 light output of Pedalite does not replace existing cycle safety lighting, but supplements it and also provides a unique light 'signature' (lights moving up and down) that helps motorists judge their distance from the cyclist.


6


7


8


9


10

**Featured Manufacturer**

**ENL**
www.enl.co.uk

# Moldflow analysis

Prior to manufacturing, Moldflow software is used to analyse and maximize the efficiency of a design. The software is suitable for all types of plastic injection molding and metal die casting. It brings together part design, material selection, mold design and processing conditions to determine the manufacturability of the part. This reduces the costs and time delays associated with otherwise unforeseen manufacturing problems. It also maximizes the efficiency of production and can reduce material consumption with significant savings.

A 3D model of the required part is generated in a suitable computer aided design (CAD) or computer aided engineering (CAE) software package.

Moldflow is a predictive analysis tool used to simulate the 3D model in production to analyse filling, packing and cooling.

The examples below illustrate analysis of flow, warp, fibre orientation, cooling and stress.

**MPI/FLOW**

MPI/Flow simulates the filling and packing phases in the molding process, helping to predict the behaviour of the material as it flows through the die cavity. This is used to optimize the location of the gate, balance runner systems and predict potential problems. Different versions are used to simulate different plastic and metal molding techniques.

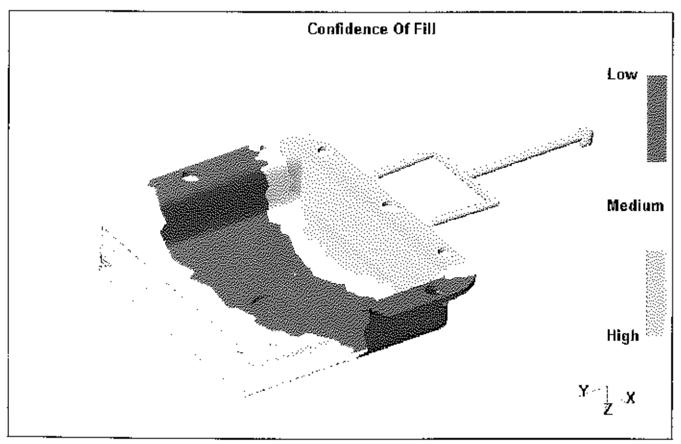
The MPI/Flow is here demonstrated on 3 products. Two stages of an Abtec part are simulated (**images 1** and **2**) to demonstrate confidence of fill, which is colour coded. MPI/Flow was used to simulate various gate positions and runner system configurations.

By changing the location of the gate on the automotive hubcap for PolyOne (**image 3**) it was possible to reduce stress and make sure there were no air traps in critical areas. The colour scale indicates bulk stress.

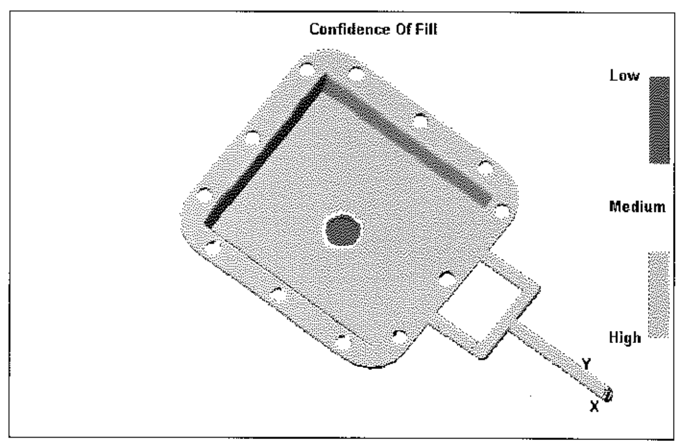
The colour scale on the automotive interior product manufactured by Resinex and Gaertner & Lang (**image 4**) indicates fill time in seconds. Appearance is very important, so the flow analysis software was used to eliminate weld lines and colour variation in critical areas. This was achieved by changing the location of the gate and temperature of the runner system.



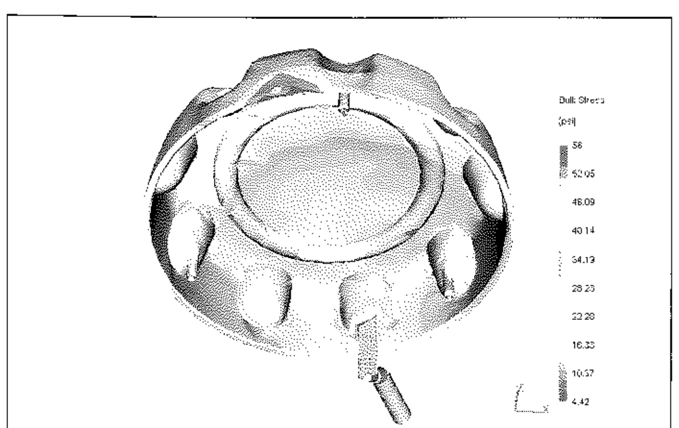


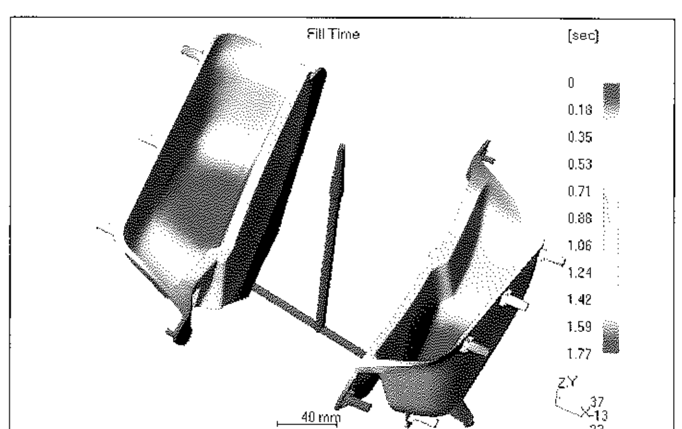


1



2


3



4

**MPI/WARP**

This analysis tool is used to predict shrinkage and warping, which are the result of stresses built up during the molding process. The information is used to specify material selection and processing parameters to minimize potential problems.

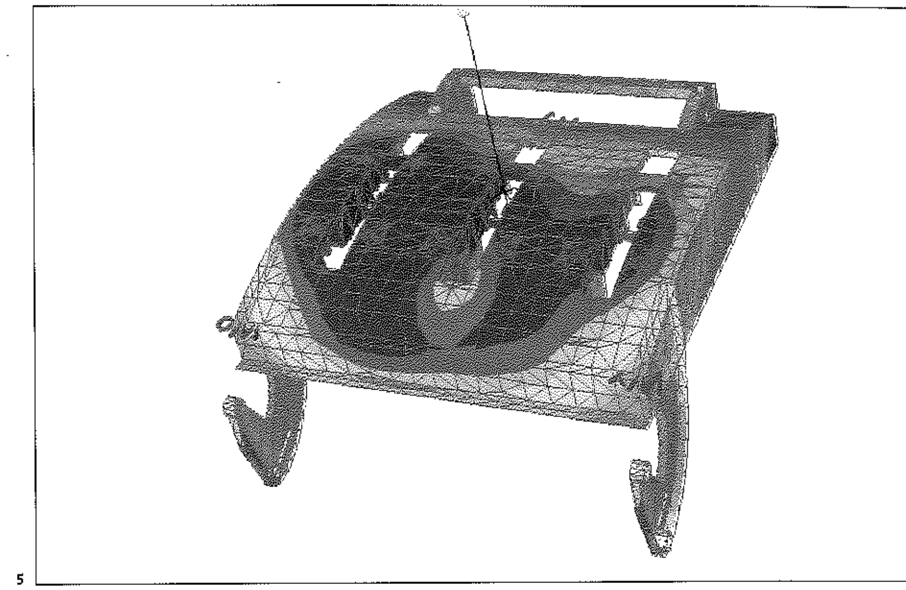
Any more than 5 mm (0.2 in.) warp was unacceptable on this Efen electronic switchboard cabin (**image 5**). The analysis found that by reducing wall thickness warp could be reduced by 90%.

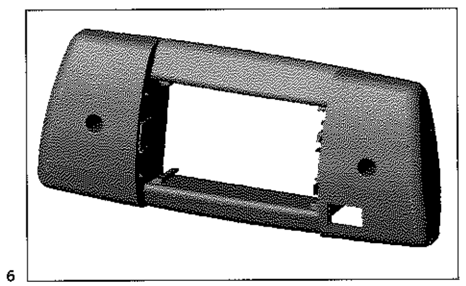
On the CAD model for a Jokon automotive lamp assembly (**image 6**), it was essential that the part did not warp so that it would maintain a water-tight seal in application. Warpage was reduced by 50% by optimizing wall thickness (**images 7** and **8**).

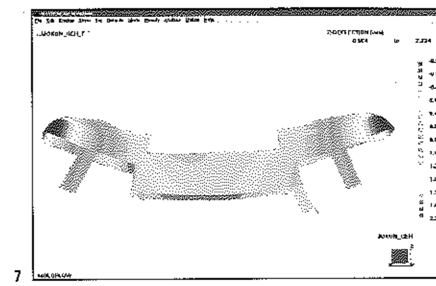
**MPI/COOL**

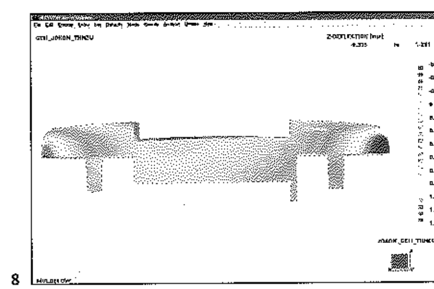
MPI/Cool is used to analyse the design of mold cooling circuits. Uniform cooling is important to make sure that the part does not warp and to minimize cycle times.

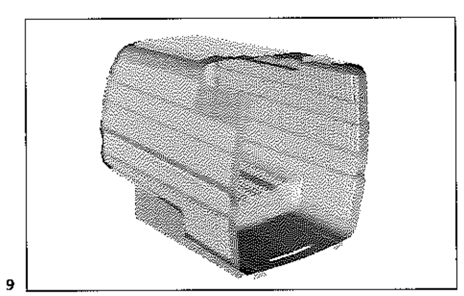
A filter housing manufactured by Hozelock shows the configuration of the mold cooling circuits (**images 9** to **11**). Changing the layout of the circuit reduced cycle time by 2 seconds and so saved more than 4% of the production cost; reducing the wall thickness reduced cycle time by 7.3 seconds and shot weight by 19.6%, saving 24% of the production costs. Combining the 2 produced 26.1% overall savings.

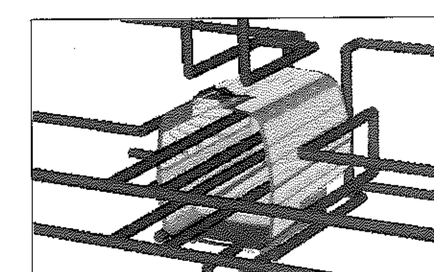

5

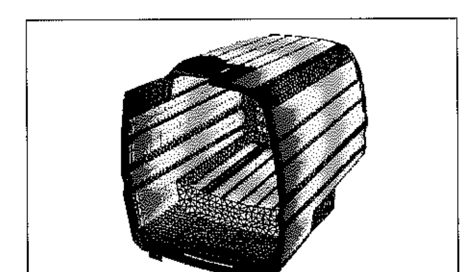

6


7


8


9


10


11

**Featured Manufacturer**

**Moldflow**
www.moldflow.com

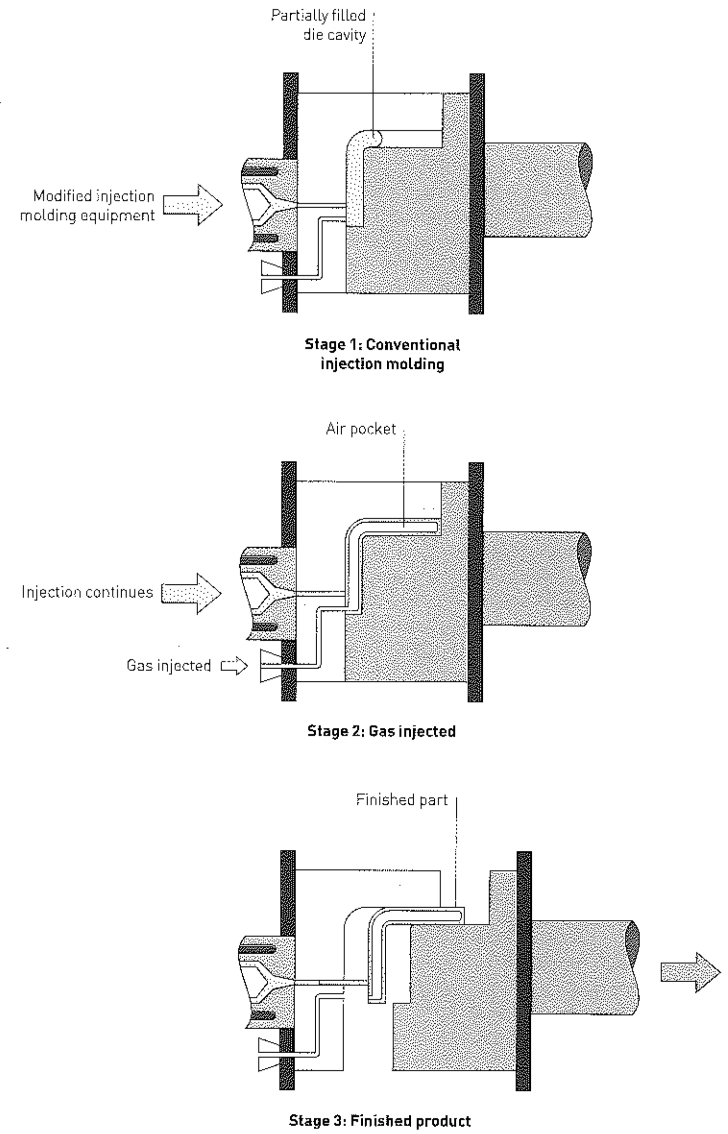
**TECHNICAL DESCRIPTION**

Gas-assisted injection molding techniques were first used in mass production in 1985. Since then the technology has steadily improved and is now into the third generation of development. Initially it was developed to overcome the problem of sink marks caused by shrinking. A small amount of gas was blown in during the injection cycle to apply internal pressure as the polymer cooled before the tool opened. With very precise computer control, it is now possible to gas fill long and complex moldings. Each cycle will be slightly different because the computer makes adjustments for slight changes in material properties and flow.

The process uses modified injection molding equipment. In stage 1, plastic is injected into the mold cavity but does not completely fill it. In stage 2, gas is injected, which forms a bubble in the molten plastic and forces it into the extremities of the mold. The plastic and gas injection cycles overlap. This produces a more even wall thickness because as more plastic is injected the air pressure pushes it through the mold like a viscous bubble. The gas bubble maintains equal pressure even over long and narrow profiles. Wall thickness can be 3 mm [0.118 in.] or more.

In stage 3, as the plastic cools and solidifies, the gas pressure is maintained. This minimizes shrinkage. Less pressure is applied to the plastic because the gas assists its flow around the die cavity.

## Gas-Assisted Injection Molding Process




1

Case Study

## Gas injection molding the Magis Air Chair

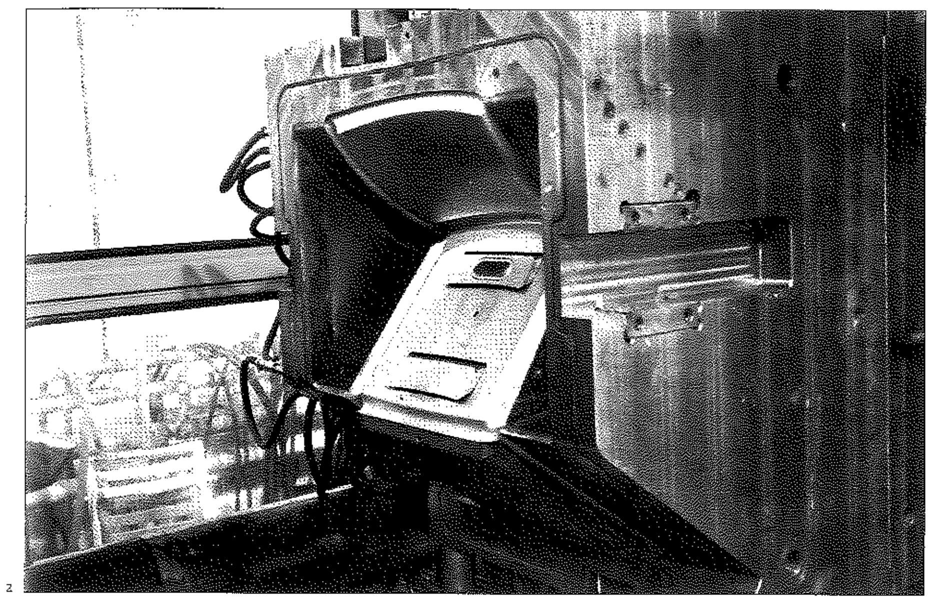
The Air Chair was designed by Jasper Morrison and production began in 2000 (**image 1**). The gas injection molding sequence takes approximately 3 minutes (**images 2–5**).

The sample cut from the leg of the Air Table shows 2 technologies (**image 6**). The first is gas injection, which creates the hollow profile. The second technology is the thin skin around the outside of the material: there is a clear division between the outer unfilled PP and the glass filled structural internal PP.

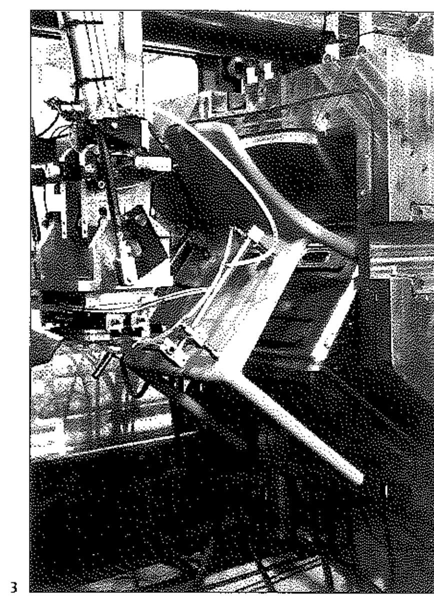
Two materials are used because the outer skin is aesthetic and therefore should not be filled. However, unfilled PP is not strong enough to make the entire structure.

The 2 layers of material in this sample are produced in a similar way to gas injection. The outer skin is injected first. The glass filled PP is injected behind it in a technique known as 'packing out'. The second material acts like a bubble of air and pushes the first material further into the die cavity without breaching it. Finally, the gas is injected to produce a hollow section and make it rigid but lightweight.

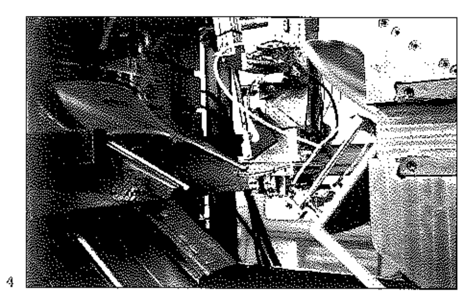
The gas injection molding technique produces a plastic chair with a very good surface finish. It weighs only 4.5 kg (9.92 lb) and is capable of withstanding heavy use.



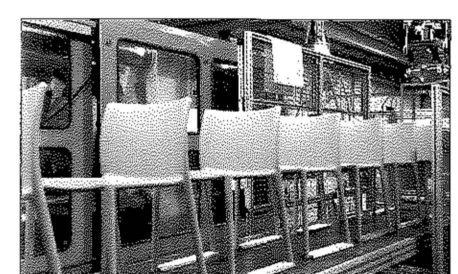
2



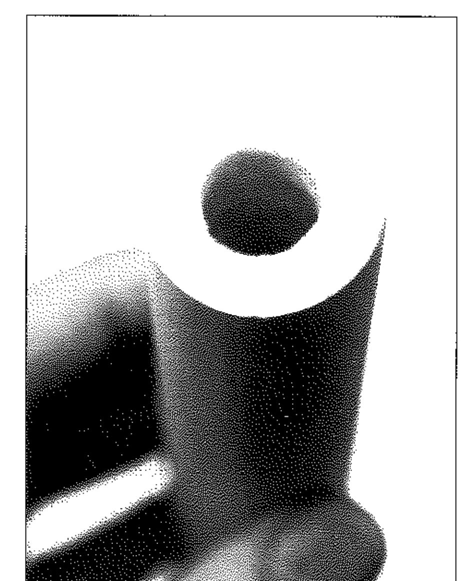
3



4



5

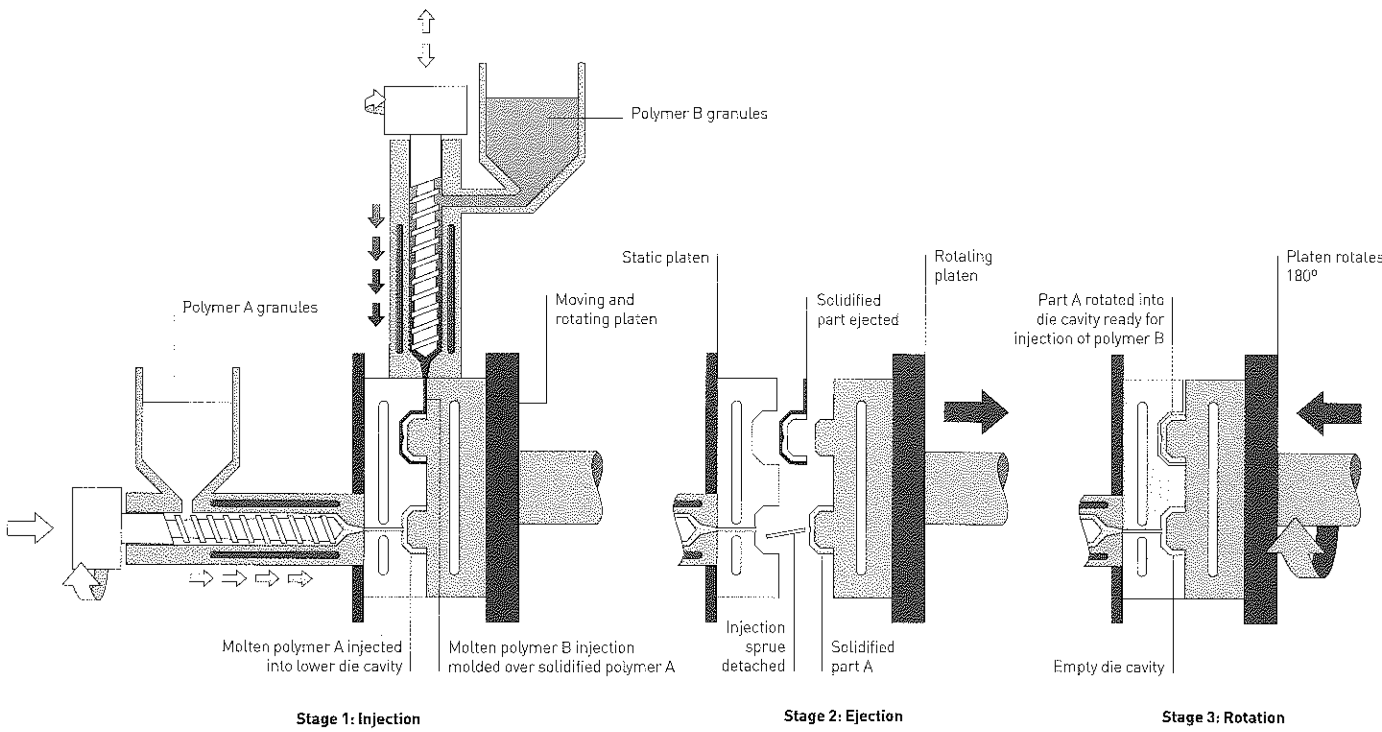


6

**Featured Company**

**Magis**
www.magisdesign.com

## Multishot Injection Molding Process




### TECHNICAL DESCRIPTION

**Injection molding 2 or more plastics together is known as multishot or overmolding. The difference is that multishot is carried out in the same tool. Overmolding is a term used to describe injection molding over any preformed material, including another thermoplastic, or metal insert, for example.**

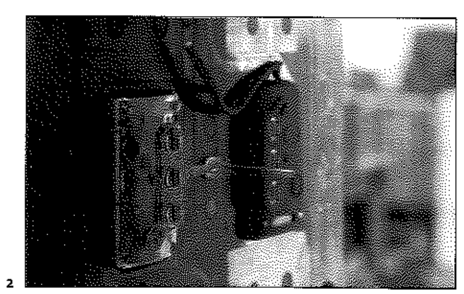
**The process of multishot injection molding uses conventional injection molding machines. It is possible to multishot up to 6 different materials simultaneously, each one into a different die cavity in the same tool.**

**The tool is made up of 2 halves: one is mounted onto a static platen, the other onto a rotating platen. Like conventional injection, this process can have complex cores, inserts and other features.**

**In stage 1, polymers A and B are injected at the same time into different die cavities: polymer A is injected into the lower die cavity; meanwhile, polymer B is injected over a previously molded polymer A in the upper cavity. The molten polymers form a strong bond because they are fused together under pressure.**

**In stage 2, the molds separate and the sprue is removed from molded polymer A. Meanwhile the finished molding is ejected from the upper die cavity. The rotating platen spins to align molded polymer A with the upper die cavity. In stage 3, the mold closes again and the sequence of operations is repeated.**

1


2



Case Study

## Multishot injection molding a handheld gas detector

This product is molded by Hymid for Crowcon. It is a handheld gas detection unit (**image 1**). Multishot injection molding has very important benefits that are essential for the effectiveness of this device, which is potentially lifesaving. The part is made up of a water clear polycarbonate (PC) body and thermoplastic electrometric (TPE) covering. The features of these materials are combined by multishot injection molding.

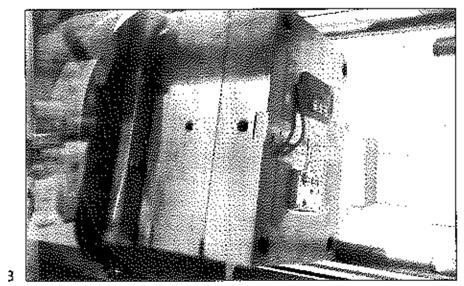
This is a tricky combination because the materials operate at different temperatures. Therefore, the runner system for the PC is heated with oil, whereas the TPE runner is cooled with water.

Of the 2 die cavities (**image 2**), the closest has just been injected with water clear PC. This gives the product rigidity, toughness and impact resistance. The farthest die cavity is the PC with a TPE molded over it. The TPE provides integral features with hermetic seals, such as flexible buttons and a seal between the 2 halves.

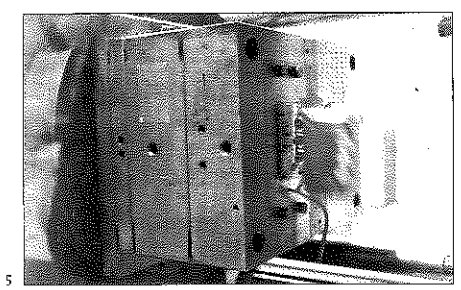
The mold half with the moldings rotates through 180° (**images 3** and **4**). In doing so, it brings the solidified PC into alignment with the second injection cavity. Then the finished molding is ejected (**image 5**) ready for the next injection cycle to commence.

The knurled metal inserts (**image 6**) are incorporated in the PC by overmolding. These are inserted into the mold by hand prior to each injection cycle.

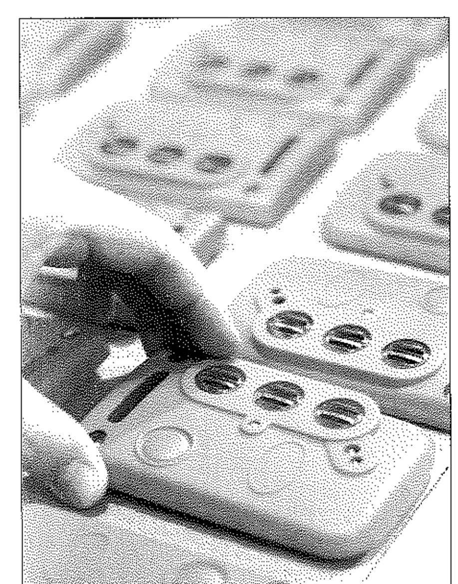
The finished moldings are stacked (**image 7**) ready for assembly. The integral and flexible button detail is shown in the final product (**image 8**).

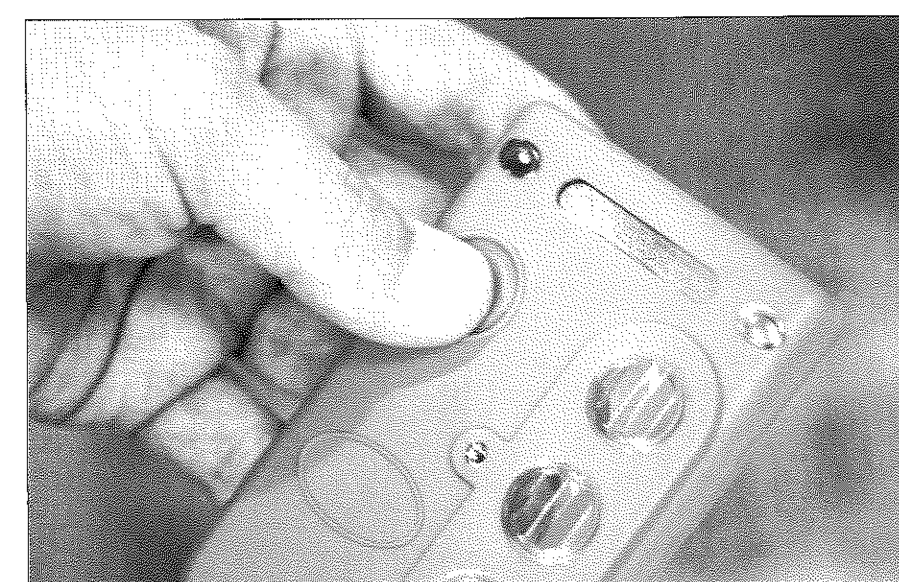

3


4


5


6


7


8

**Featured Manufacturer**

**Hymid Multi-Shot**
www.hymid.co.uk

**TECHNICAL DESCRIPTION**

The in-mold decoration process is used to apply print to plastic products during injection molding, thus eliminating secondary operations such as printing and spraying. However, the cycle time of injection molding is increased slightly. The process is used in the production of nearly every modern mobile phone, camera and other small injection molded product.

In stage 1, a printed PC film is loaded into the die cavity prior to injection molding. The print side is placed inwards, so that when it is injection molded the print will be protected behind a thin film of PC.

In stage 2, when the hot plastic is injected in the die cavity, it bonds with the PC film. (This is similar to multishot injection molding.) In stage 3, the film becomes integral with the injection molded plastic and has a seamless finish, with a printed surface.

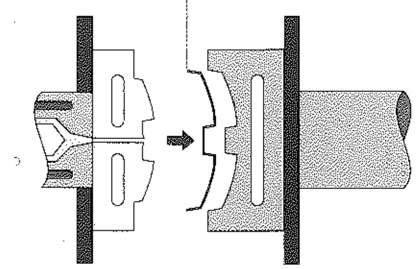
If the surface of the mold is not flat or slightly curved then the film is thermoformed to fit exactly. When the hot plastic is injected it is forced against the mold face at pressures between 30 and 17,000 N/cm² (20–11,720 psi). The pressure is determined by the type of material and surface finish.

Another technique, known as insert film molding, differs because the film is supplied as a continuous sheet. It is sucked into the die cavity by a strong vacuum (similar to the thermoforming process), the mold closes and the injection process follows.

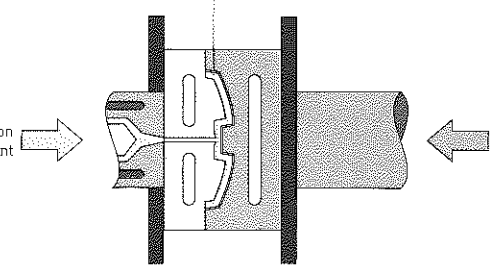
**Featured Company**

**Luceplan**
www.Luceplan.com

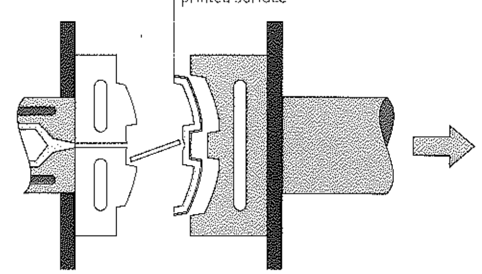
## In-Mold Decoration Process



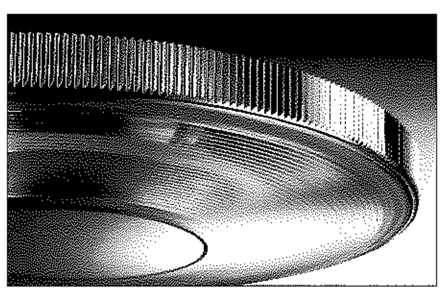

**Stage 1: Film inserted**

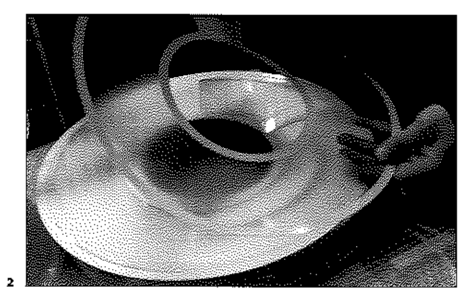


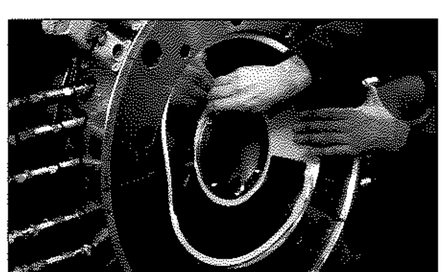

**Stage 2: Conventional injection molding**




**Stage 3: Finished product**


1


2


3



Case Study

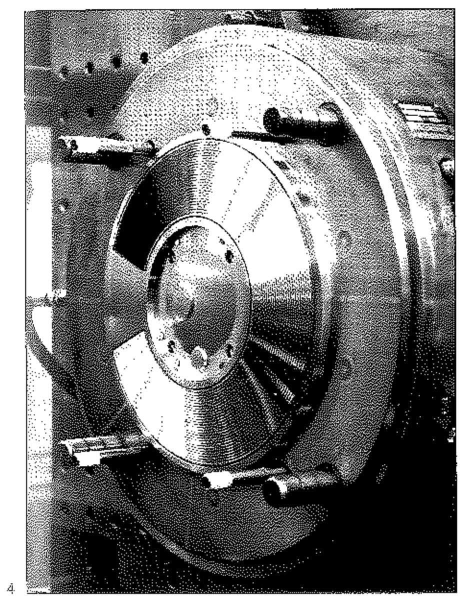
## In-mold decoration Luceplan Lightdisc

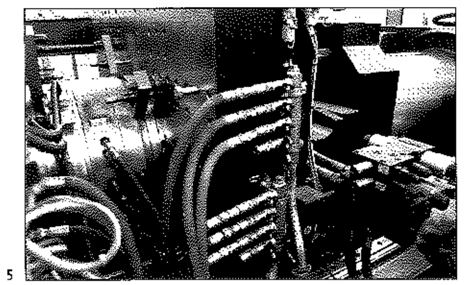
This case study demonstrates the production of the Luceplan Lightdisc (**image 1**). It was designed by Alberto Meda and Paolo Rizzatto in 2002. By incorporating in-mold decoration, the diffuser acts as shade, too. Graphics and instructions are included on the in-mold film and this eliminates all secondary printing.

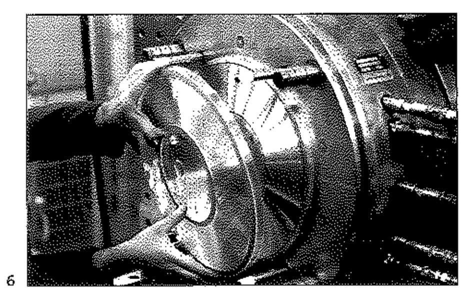
The process is similar to conventional injection molding, except that a printed film is placed into the die cavity.

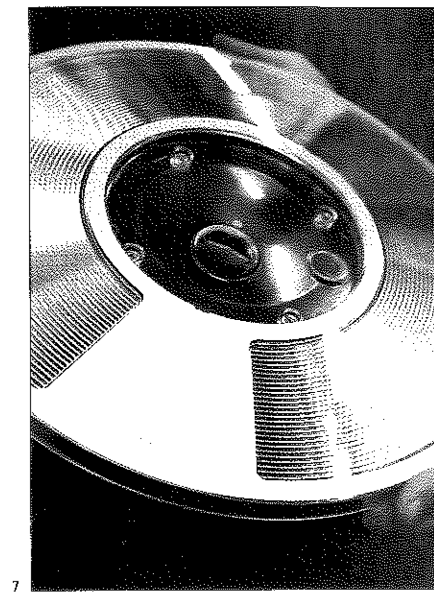
The film is prepared and placed into the mold by hand (**images 2** and **3**). The opposite side of the mold is textured to provide the diffusing effect in the light (**image 4**). The mold is clamped shut under 600 tonnes (661 US tons) of hydraulic pressure and the injection molding takes place (**image 5**). The part is demolded by hand (**image 6**), but this can also be carried out using a robot.

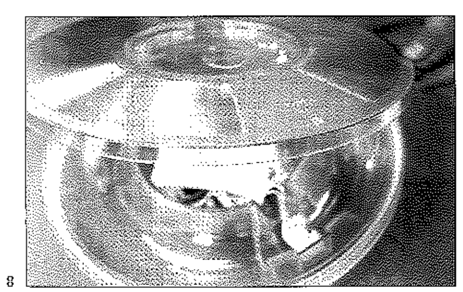
This finished molding is inspected prior to assembly (**image 7**). Screw bosses are incorporated into the injection molding, so the assembly procedure is relatively straightforward. The electrics are inserted and fixed in place (**image 8**). Then the 2 halves of the Lightdisc are screwed together (**image 9**). The fixings are covered with a snap fit enclosure (**image 10**), which means the whole product can be taken apart for maintenance and recycling.

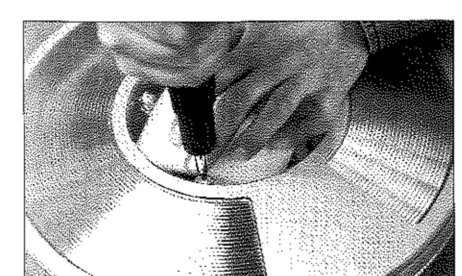

4

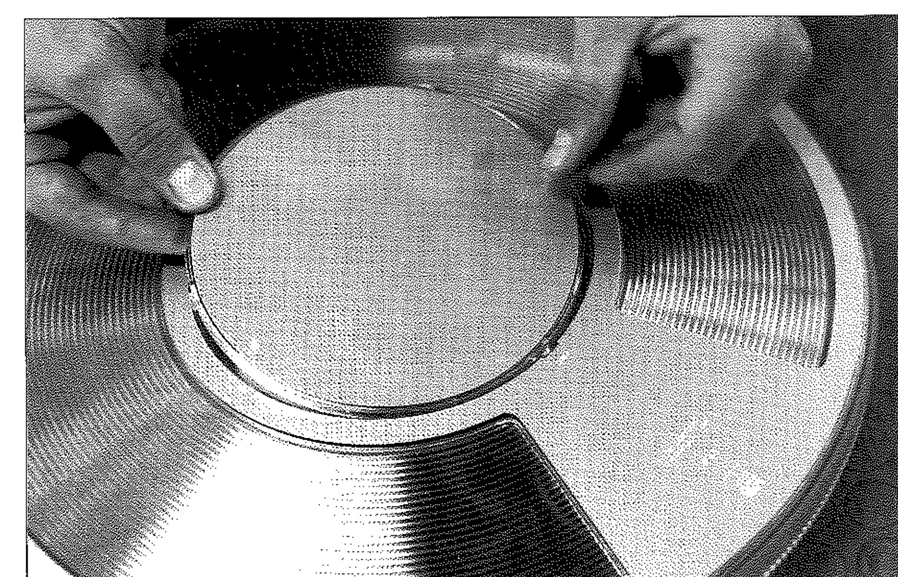

5


6


7


8


9


10