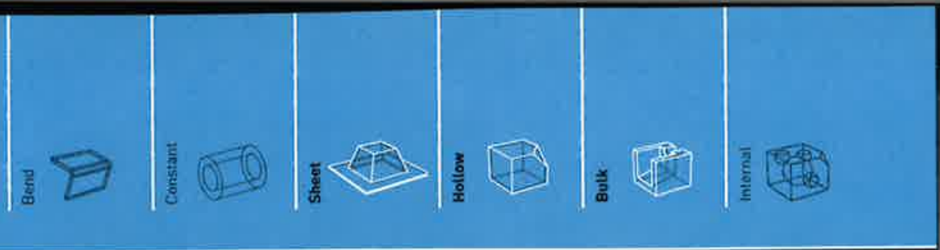
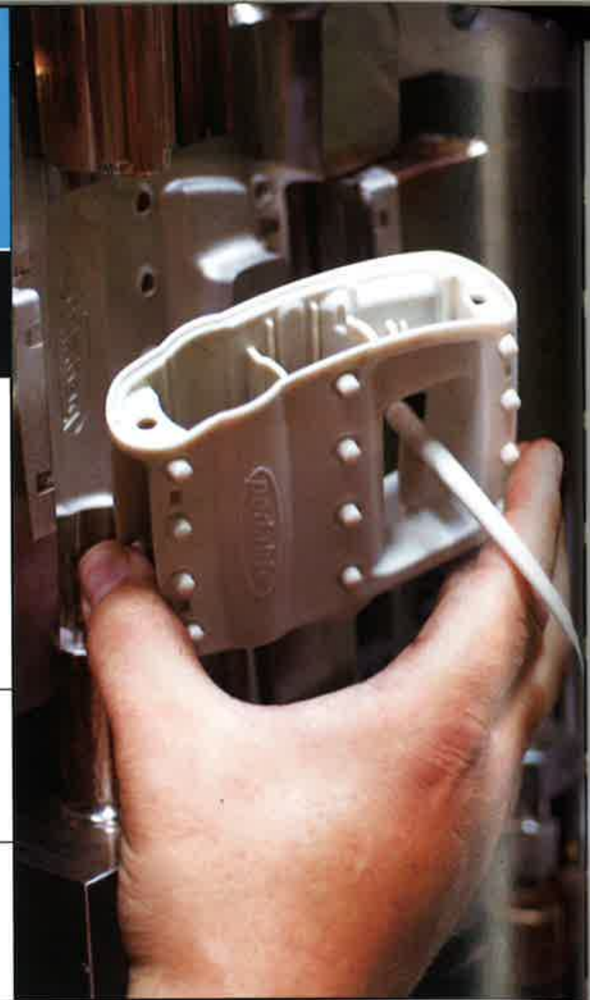


Injection Molding



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 • Very low unit costs	• Automotive • Consumer electronics and appliances • Industrial and household products	• High volume mass production
Quality	**Related Processes**	**Speed**
• Very high surface finish • Highly repeatable process	• Reaction injection molding • Thermoforming • 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.

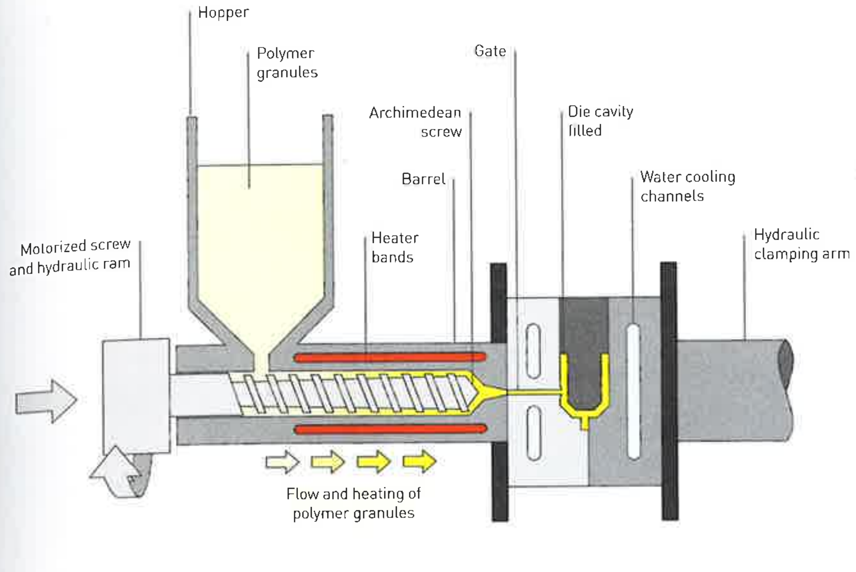
QUALITY

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

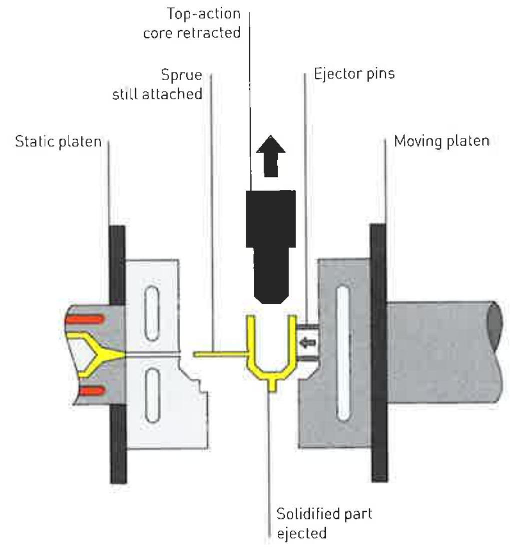
The downside of the high pressure is that the resolidified polymer has a tendency to shrink and warp. These defects can be designed out using rib details and careful flow analysis.

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).

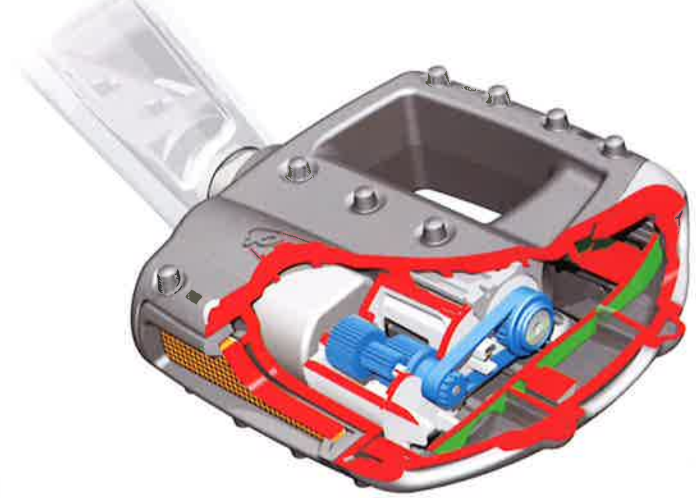
A close working partnership with ENL ensured Product Partners' ideas were successfully translated through the design, development and toolmaking process, ensuring a 'right first time' product. The technically challenging combined overmold assembly, for instance, was approved at the first-off tool trials.

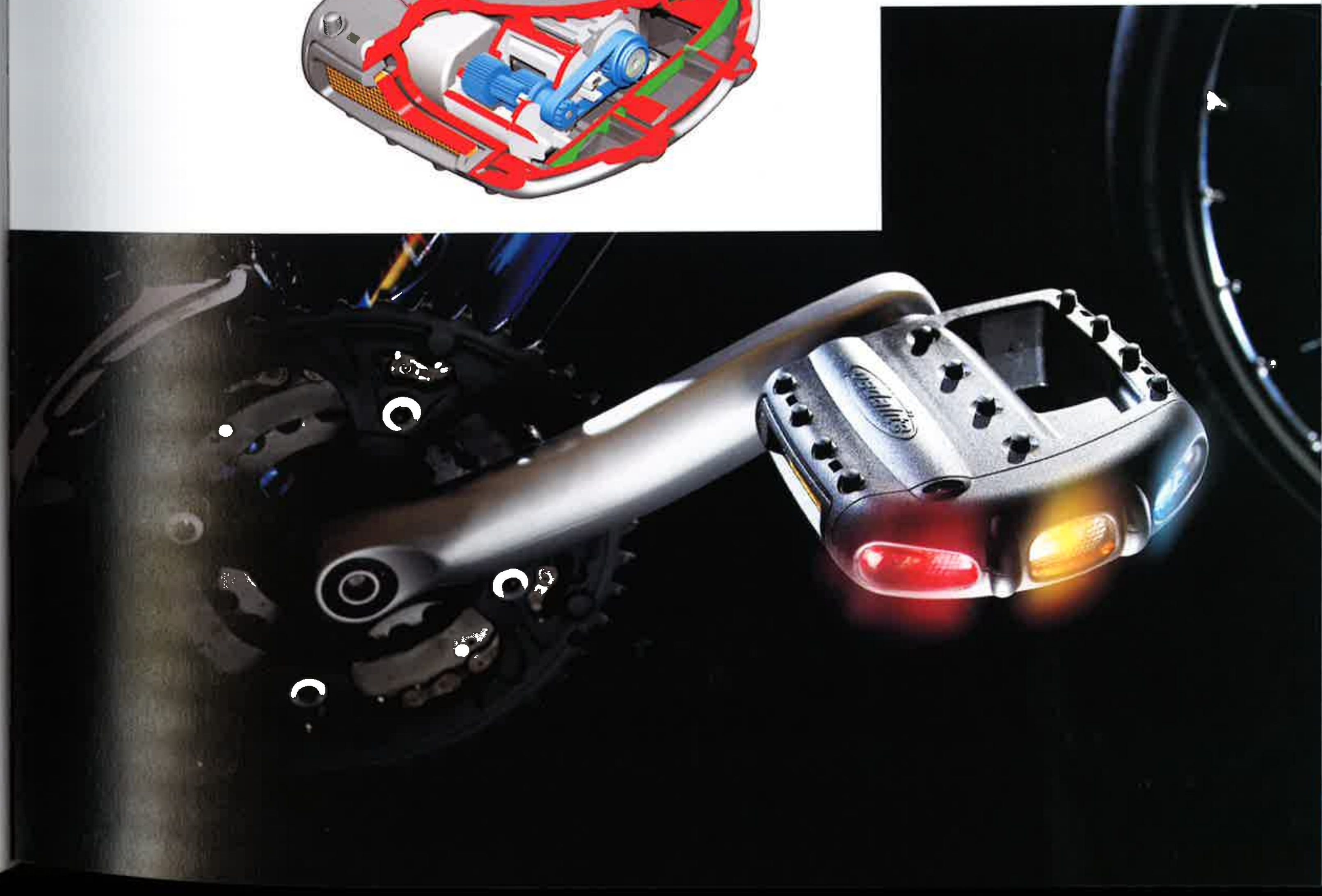
To ensure the gearbox concept was feasible Davall Gears were recruited to provide expertise in gear ratios, gear detail design, materials specification and manufacture. Polyamide (PA) nylon was chosen for its exceptional wear characteristics and self-lubrication properties.

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

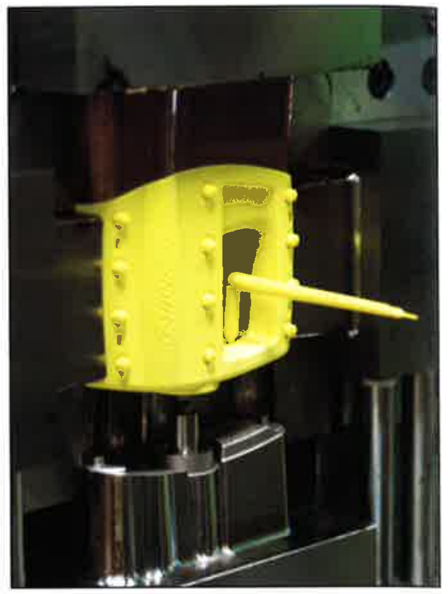
The raw material is a glass-filled nylon that is white in its raw state. If a colour is required, then pigment is added. In this case a small quantity of Clariant masterbatch yellow was used (**image 1**). The end result is a surprisingly vivid yellow colouring.

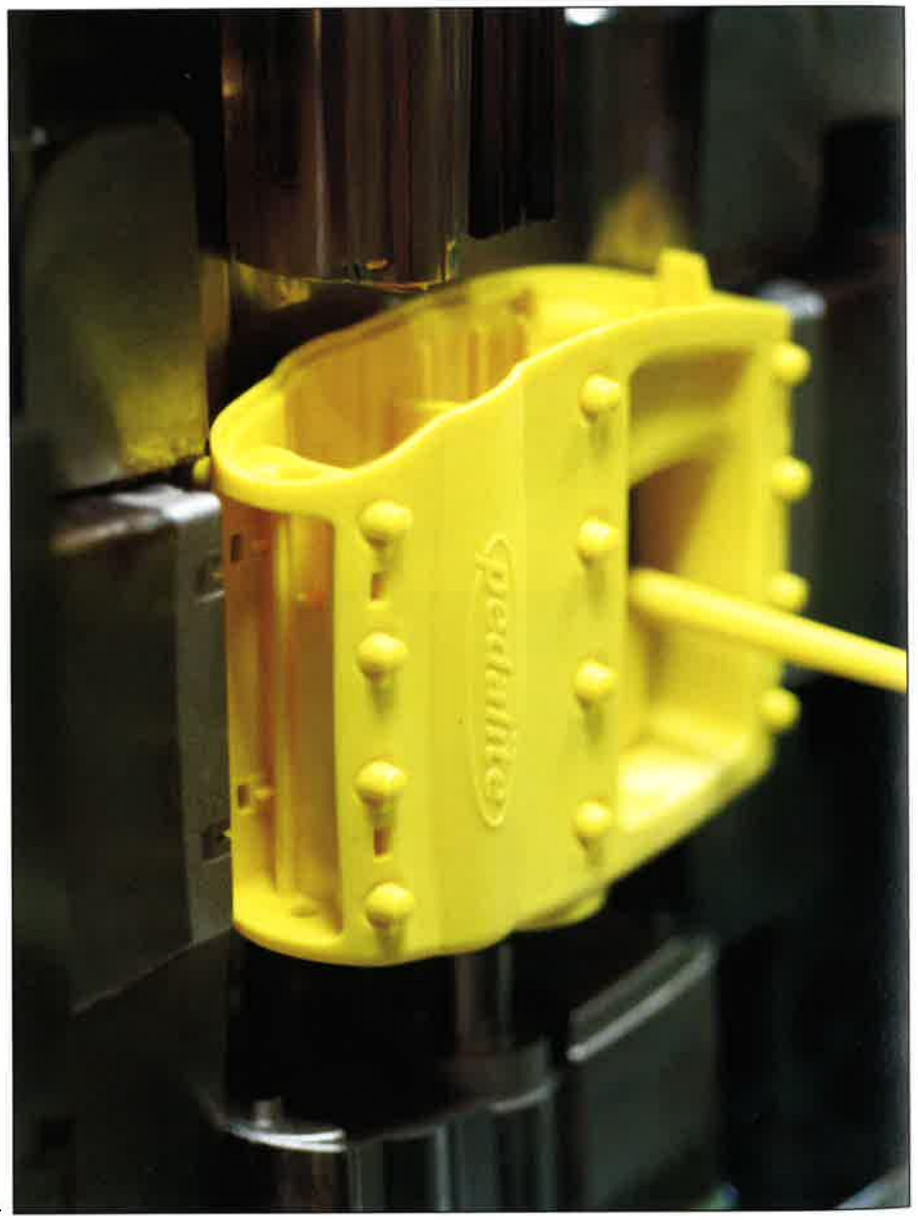
In normal operations, the injection molding process takes place behind a screen within the machine (**image 2**), but for the purposes of this case study the dies are shown in close-up with the screen open.

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