INJECTION MOULDING

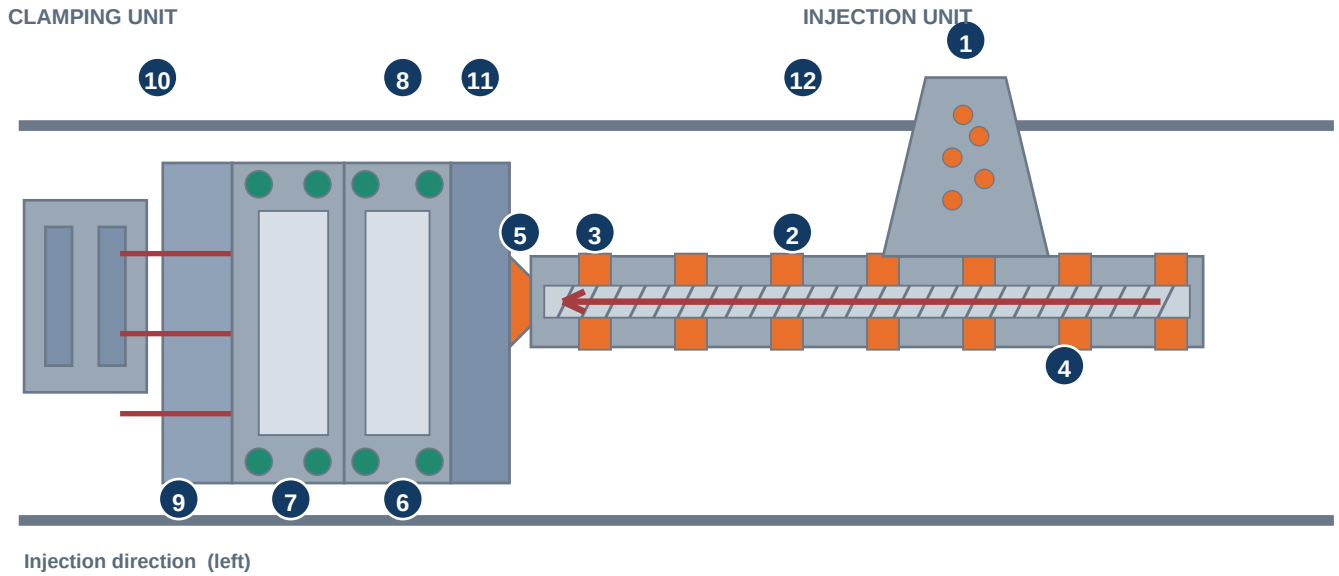
The complete manufacturing reference — process · machine · materials · design rules · defects · applications

Injection moulding is a manufacturing process for mass-producing identical plastic parts. Polymer granules are melted inside a heated barrel, then forced under very high pressure through a nozzle into a closed steel mould. The plastic fills the hollow **cavity**, cools and solidifies, and the mould opens to eject the finished part. One cycle takes only seconds, which is why injection moulding makes the billions of identical parts we use every day — bottle caps, phone cases, LEGO bricks, gears and medical syringes. It is the most important process in the IGCSE DT syllabus for **thermoplastics in high volume**.

Cycle time	**Injection pressure**	**Melt temperature**	**Mould tooling cost**
15 – 60 s per part (up to 1,000+/hr)	up to 1,500 bar (~15,000 N/cm²)	180 – 300 °C (polymer dependent)	£10k – £250k+ (amortised over volume)

HOW THE MACHINE WORKS — THE INJECTION MOULDING MACHINE

A standard machine has two units. The **injection unit** melts and pumps the plastic; the **clamping unit** holds the mould shut against the enormous pressure of the injected melt. Numbers match the table below.



1	Hopper	Holds polymer granules and feeds them by gravity into the barrel.
		Heated cylinder in which the granules melt as they travel along.
3	Heater bands	Electric band heaters around the barrel control the melt temperature zone by zone.
		Rotates to melt, mix and push plastic forward; then slides forward like a plunger to inject the melt.
5	Nozzle	Seals the barrel against the mould's sprue bush and delivers the melt under pressure.
		Tapered entry channel that takes the melt from the nozzle into the mould.
7	Mould cavity	The hollow, mirror-image space that forms the outside shape of the part.
		Water passages machined in the mould plates that remove heat so the part solidifies.
9	Ejector pins	Steel pins pushed forward to knock the cooled part off the mould.
		Hydraulic (or toggle) clamp slides the moving half of the mould open and closed and locks it under multi-tonne force.
11	Fixed platen	Stationary steel plate carrying the fixed half of the mould and the nozzle.
		Four hardened steel bars that guide the platens and absorb the clamping load.

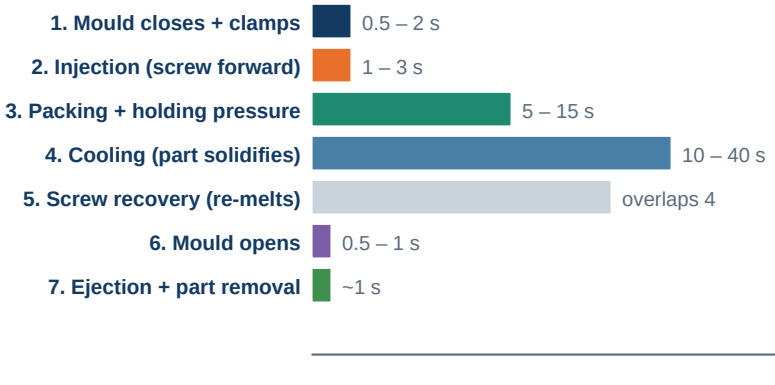
Note — thermoplastics vs thermosets

Standard injection moulding is for **thermoplastics**: the barrel heats the plastic to melt it and the *cool* mould re-solidifies it, so the cycle can repeat in seconds and scrap/runners can be re-ground and reused. **Thermosets** (e.g. epoxy, phenolic) and **elastomers** (silicone, rubber) can also be injection moulded but use a *hot* mould that cures the material by chemical cross-linking — the cure reaction sets the part, and it cannot be re-melted afterwards.

THE INJECTION CYCLE — STEP BY STEP

Every part is made by repeating the same sequence of events. The times below are typical for a medium-sized part; large parts take longer to cool.

- **Mould closes and clamps.** The moving platen slides the mould shut and the clamp locks it with enough tonnage to resist the injection pressure (otherwise melt escapes as *flash*).
- **Injection.** The screw stops rotating and acts as a ram: it shoves forward, forcing melt through the nozzle, sprue, runners and gate into the cavity — typically in 1–3 s.
- **Packing and holding.** Extra pressure is maintained for several seconds so that more melt is squeezed in to replace the volume lost as the plastic shrinks while cooling. Skipping this step gives *sink marks*.
- **Cooling.** Heat conducts from the melt into the water-cooled mould plates. The part solidifies in the cavity. This is the longest phase (10–40 s+).
- **Screw recovery (plasticising).** While the part cools, the screw rotates again, melting the next shot of granules and pulling back to the start position — it overlaps the cooling phase, which is why the cycle is so fast.
- **Mould opens.** The clamp pulls the moving half away; the part sticks to the core (male) side.
- **Ejection.** Ejector pins push the part off the core. A robot arm or gravity removes it, and the cycle repeats — shot after shot, all day.



Why clamping force matters (tonnage)

Cavity pressure × projected area of the part = the force trying to blow the mould open. The clamp must exceed this — a small machine needs ~30–100 tonnes, car-bumper machines several thousand tonnes. Too little clamp = flash; too much = wasted energy.

GATES — WHERE THE MELT ENTERS THE CAVITY

The gate is the last, narrowest part of the feed system. It controls how fast the melt enters, freezes off quickly after injection (sealing the cavity for packing), and leaves a small witness mark that is trimmed or hidden on a non-visible surface.

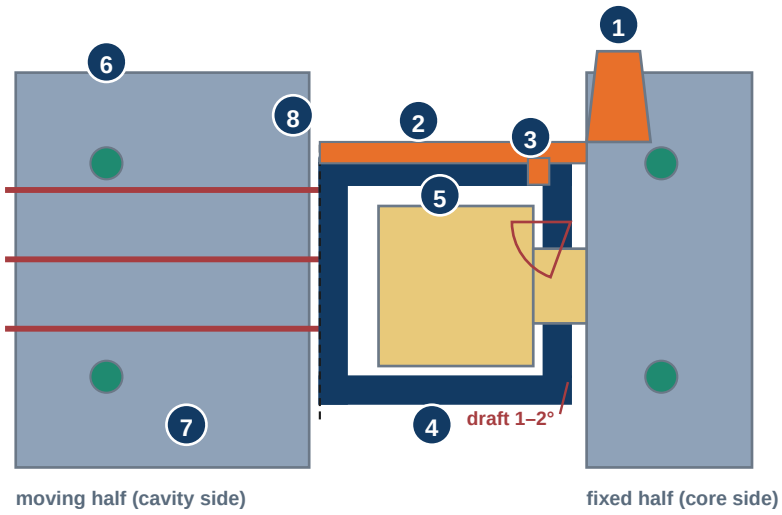
Gate type	Shape / position	Typical use
	Rectangular slit at the parting line on a side wall.	Most common; simple to machine, easily trimmed.
Pin (point) gate	Tiny round gate that breaks off cleanly.	Multi-cavity and 3-plate moulds; small parts.
	Tunnel gate under the parting line.	Automatic de-gating; high production rates.
Fan gate	Wide, flared entry spreading the flow.	Large flat parts; reduces stress and flow marks.
	Circular gate around the whole end of the part.	Hollow cylindrical parts (pipes, bobbins) — even fill.
Hot runner	No cold sprue/runner: the feed stays molten inside the tool.	High volumes — zero runner waste, faster cycles.

Gate position matters

Gates should feed the **thickest section** first so the melt can pack shrinkage effectively. Poor gate placement causes weld lines, orientation effects, jetting and witness marks — gate design is one of the first things a mould designer decides.

ANATOMY OF THE MOULD (THE TOOL)

The mould is a precision steel tool, usually in two halves. It is the most expensive part of the whole process — often £10,000–£250,000 — because every feature of the part is machined into it to mirror-image accuracy.



1	Sprue bush	Tapered cone that receives melt from the nozzle and feeds the runner system.
		Channel that distributes melt to every cavity. Should be balanced so all cavities fill at the same time.
3	Gate	Restricted entry to the cavity — controls fill rate, freezes first, is trimmed from the part.
		Machined hollow that forms the outside of the part. Its surface finish becomes the part's finish.
5	Core (male)	Protruding block that forms the inside/hollow of the part; the part shrinks onto it on cooling.
		Water lines (often conformal) that remove heat fast and uniformly — cooling time dominates the cycle.
7	Ejector pins	Push the part off the core once the mould opens.

1	Sprue bush	Tapered cone that receives melt from the nozzle and feeds the runner system.
		Where the two halves meet. It must be positioned so the part can release; a witness line is left on the part.

MOULDED-IN FEATURES — OVERMOULDING & INSERT MOULDING

- **Insert moulding:** a metal insert (threaded boss, nut, terminal) is placed in the cavity before injection; the plastic moulds around it, locking it permanently into the part. Used for electrical plugs, hinges, medical valves.
- **Overmoulding:** a second material is moulded over a first part — e.g. soft TPE grip over a hard ABS handle (toothbrushes, power tools). Done in a second shot or a two-shot machine.
- **In-mould labelling:** a printed label is placed in the cavity; the melt fuses to it, giving permanent decoration with no post-printing (food tubs, bottle crates).

Undercuts

Any feature that locks the part into the mould — an internal lip, a hole at right angles to the pull direction, a snap-fit barb — is an **undercut**. Undercuts need moving parts in the tool (**slides** for outside features, **lifters** for inside ones), which add cost and cycle time. Good designers avoid them or re-orient the part so they disappear.

MATERIALS — WHICH POLYMER FOR WHICH PART?

Injection moulding works with most thermoplastics — each has a different melt temperature, shrinkage and mechanical personality. The mould is made for one material, so the polymer choice must come **before** the tool is cut.

Polymer	Key properties	Typical moulded products
	Light, tough, chemical-resistant, excellent fatigue life; cheap.	Bottle caps, food containers, living hinges, car bumpers, crates.
PE — Polyethylene (HDPE/LDPE)	Waxy feel, tough, flexible, very cheap, good chemical resistance.	Bottles, toys, buckets, storage crates, lids, fuel tanks.
	Rigid, glossy, cheap but brittle; easy to mould precisely.	CD cases, cutlery, yoghurt pots, model kits, disposable razors.
ABS	Tough, impact-resistant, good surface finish, paints well.	LEGO bricks, phone cases, power-tool housings, dashboards, luggage.
	Rigid or flexible (with plasticiser); flame-resistant, durable.	Fittings, cable plugs, pipe connectors, shoe soles.
PA — Polyamide (nylon)	Strong, stiff, low friction, wear- and heat-resistant.	Gears, bearings, clips, zip fasteners, under-bonnet car parts.
	Very stiff, low friction, holds tight tolerances.	Precision gears, sliding parts, pump components.
PC — Polycarbonate	Transparent, very strong and impact-resistant, heat-resistant.	Safety goggles, lenses, headlamp covers, bullet-resistant glazing.
	Crystal-clear, UV-stable, scratches easily.	Lenses, light covers, display stands, car tail-lights.
PET — Polyester	Clear, tough, food-safe, good barrier.	Drink-bottle preforms, food trays, jars.

ADDITIVES & COMPOUNDS

Additive	What it does
Glass fibre (10–40%)	Hugely increases strength and stiffness, raises heat resistance, reduces shrinkage — but wears the mould and shrinks anisotropically.
Talc / chalk filler	Lowers cost and increases stiffness; makes surfaces matt.
Colour masterbatch	Concentrated pigment pellets added at the machine — one material can make any colour.
UV stabilisers	Stop outdoor parts fading or becoming brittle in sunlight.

Additive	What it does
Flame retardants	Make parts self-extinguishing for electrical housings (UL94 ratings).
Plasticisers	Softened PVC from rigid pipe to flexible cable insulation.
Release agents	Reduce sticking so parts eject cleanly.

Choosing a material — decision flow

Mechanical loads (strength/stiffness/impact) - service temperature & chemical environment - appearance and finish - cost per part and volume - safety rules (food contact, medical, flammability) - recyclability. For a cheap, tough, food-safe everyday part the answer is usually PP or PE; for a precision gear, PA or POM; for an impact shell, ABS or PC.

DESIGN FOR MANUFACTURE — RULES THAT SAVE MONEY

Rule	Why	Guide values
Keep wall thickness uniform	Thick sections cool slower - sink marks, voids and warpage. Transitions must be gradual.	Typical 1–4 mm; keep within ±25% across the part.
Use ribs instead of thick walls	A rib adds stiffness without adding heavy, sink-prone mass.	Rib base ~50–60% of wall thickness; draft the sides.
Add draft angle to vertical faces	The part shrinks onto the core; without taper it will not release.	0.5–2°; more (2–5°) for textured surfaces.
Radius every corner	Sharp corners concentrate stress and create weak, cracked parts.	Inside radius at least 25–50% of wall thickness.
Design bosses for screws	Bosses locate and hold self-tapping screws without extra parts.	OD ~ 2x screw dia.; boss wall ~60% of part wall.
Plan for shrinkage	All polymers shrink 0.5–2.5% as they cool; the cavity is cut oversize to compensate.	Semi-crystalline (PP, PA) shrink most; amorphous (ABS, PC) least.
Avoid undercuts	They need slides/lifters — more tool cost, slower cycle.	Re-orient the part or open up the feature.
Position gates at thick sections	Melt must pack the thickest volume while the gate is still open.	Hide witness marks on non-visible faces.
Design cooling symmetry	Uneven cooling = uneven shrinkage = warpage.	Balance steel mass and water lines around the cavity.
Think about tolerances	Tight tolerances need premium tooling and slower, more expensive moulding.	±0.1–0.5 mm is normal; ±0.02 mm is precision work.

SURFACE TEXTURE, LETTERING & LIVE HINGES

- **Texture:** the cavity is chemically etched or spark-eroded, so texture is moulded in — no painting needed. Matte finishes hide sink marks better than gloss.
- **Lettering:** raised lettering on the part is cut *into* the cavity (easier to machine and polish); never place lettering across the parting line — it will be sheared.
- **Living hinges:** PP and PE can be moulded as one part with a thin hinge (~0.3–0.5 mm) that flexes millions of times — bottle caps and lunch boxes use this. The polymer chains must align across the hinge, so the melt must flow across it.
- **Threads:** external threads unscrew from the mould or split into dies; internal threads need a collapsible core or unscrewing device — or mould a plain hole and use an insert.

Mould steel & tool life

Moulds are made from hardened tool steel (P20, H13) or stainless for medical parts. A well-maintained mould survives 500,000 – several million shots. Worn moulds produce flash and poor surfaces — which is why tool maintenance is scheduled like an airline check.

DEFECTS — WHAT GOES WRONG & HOW TO FIX IT

Every defect has a cause in the process window (temperature / pressure / speed / timing) or in the design. Diagnose the symptom, change one variable at a time, and re-shoot.

Defect	What it looks like	Main causes	Fixes
Short shot	Part incomplete; cavity not fully filled.	Not enough material, pressure too low, gate too small, cold melt.	Increase shot size, pressure, melt & mould temperature; enlarge gate.
Flash	Thin plastic burr along the parting line.	Clamp force too low, mould worn or not shut properly, vent gap too big.	Raise clamp tonnage; repair or re-set the mould; reduce injection pressure.
Sink marks	Shallow depressions on thick sections.	Local shrinkage at thick walls; packing pressure released too early.	Uniform walls or ribs instead; hold pressure longer; cool longer.
Weld / knit lines	Faint lines where melt fronts meet around cores.	Two flow fronts meet and don't fully fuse (around holes/inserts).	Raise melt & mould temperature; move or add gates; add vents.
Warpage	Part twists or bows after ejection.	Uneven shrinkage from uneven cooling or wall thickness.	Balance cooling, uniform walls, symmetrical gate layout.
Burn marks	Black/brown scorch patches at the end of fill.	Trapped air compressed until it 'diesels' (burns); melt too hot.	Add vents at last-filled areas; lower melt temp; slow the injection.
Jetting	Squiggly worm-like streaks from the gate.	Melt shoots across the cavity and freezes before the mould fills.	Raise temperatures, enlarge gate, ramp injection speed up gradually.
Voids / bubbles	Air pockets inside thick walls.	Trapped air or shrinkage at the core of thick sections.	Thinner walls, higher hold pressure, better venting.
Discolouration	Streaks or off-colour areas.	Polymer degraded by heat, contaminated regrind, wrong pigment.	Lower temperatures, purge the barrel, clean material supply.
Sticking / ejector marks	Part won't release; white dents from pins.	No draft, undercut, insufficient cooling, too few ejectors.	Add draft, more/bigger ejector pins, longer cooling, release agent.
Flow lines	Rippled rings near the gate.	Melt cools in waves as it advances; too slow fill.	Raise melt & mould temperature; inject faster.
Gate blush	Frosty / stress marks around the gate.	High shear as melt races through a small gate.	Larger or fan gate; lower injection speed; hotter melt.

Golden rule of defect fixing

Change ONE variable at a time, log every setting (the 'process sheet'), and shoot test parts after each change. Moulding is a repeatable science — random knob-twiddling just creates a new defect.

ADVANTAGES & DISADVANTAGES

Advantages	Disadvantages
• Complex 3-D shapes with internal detail in one shot • Extremely fast cycles - hundreds to thousands of parts per hour • Excellent, moulded-in surface finish — little or no finishing work • High accuracy and repeatability (LEGO: ±0.002 mm) • Minimal material waste — runners and sprues are re-ground • Wide material choice with tailored additives • Moulded-in features: threads, inserts, living hinges, snap fits • Very low labour cost per part once running	• Very high initial tooling cost (£10k–£250k+) • Only economical for HIGH VOLUMES (usually > 10,000 parts) • Long lead time: mould design + machining takes weeks–months • Design changes are very expensive once the tool is cut • Part size limited by machine size (max. shot weight) • Mould maintenance and wear over long runs • Undercuts and tight tolerances push cost up sharply

REAL-WORLD APPLICATIONS

If it is plastic, identical, and made by the million, it was almost certainly injection moulded: **LEGO bricks** (moulded to ±0.002 mm, billions per year), **bottle caps** (single cavities run 40,000+ per hour), **medical syringes** and IV connectors

(moulded in cleanrooms), **phone and electronics housings** (ABS/PC), **car dashboards, bumpers and air vents**, **gears, bearings and clips** (PA/POM), **stacking chairs, storage boxes and bins**, toys, pens, kitchenware, power-tool housings, and packaging of every kind. If the shape is hollow and sealed (bottles, containers), the sister process blow moulding is usually used instead.

SUSTAINABILITY & ENVIRONMENT

- **Waste:** cold-runner sprues and rejects are ground into 'regrind' and blended back into new material (typically 10–30%), so scrap is close to zero.
- **Energy:** moulding uses roughly 2–5 MJ per kg of polymer. Most energy goes into melting and machine hydraulics — electric servo machines cut this by up to 60%.
- **Weight:** plastic parts are light, cutting fuel in transport; lightweight car panels reduce vehicle emissions.
- **Recycling:** recycled (PCR) resins and biopolymers (PLA, starch blends) are now regularly moulded. Design for recycling = single polymer, no bonded inserts, identifiable material marking.
- **Problem:** cheap single-use items encourage waste; mixed-material products are hard to recycle; additives complicate recovery.

COMPARED WITH OTHER POLYMER PROCESSES

Process	Part type	Complexity	Volume	Tooling cost
Injection moulding	Solid 3-D parts, any detail	High	10k – millions	Very high
Blow moulding	Hollow, sealed: bottles, tanks	Low	Very high	High
Extrusion	Continuous profiles: pipes, sheets	Low	Continuous, huge	Low – medium
Thermoforming	Thin shells: trays, blisters	Low – medium	Medium – high	Low
Vacuum forming	Shallow shells: signs, packaging	Low	Low – medium	Lowest
Compression moulding	Thick thermoset parts: electrical	Medium	Medium	Medium

GLOSSARY — KEY TERMS

Shot — One complete injection cycle; one 'dose' of melt.	**Sink mark** — Surface depression over a thick section.
Cycle time — Total time for one complete shot.	**Weld line** — Weak line where two melt fronts meet.
Sprue / runner / gate — The feed system: main channel, distribution channels, and entry into the cavity.	**Undercut** — Feature that traps the part in the mould.
Cavity / core — Female hollow and male block that together form the part.	**Side action / slide** — Moving tool part that forms an outside undercut.
Parting line — The meeting surface of the two mould halves.	**Ejector pins** — Pins that push the finished part out.
Draft angle — Taper on vertical walls so the part releases.	**Regrind** — Ground-up sprues/runners/rejects reused in the mix.
Shrinkage — Contraction of the polymer as it cools (0.5–2.5%).	**Masterbatch** — Concentrated colour/additive pellets.
Tonnage — Clamping force of the machine, in tonnes.	**Overmoulding** — Moulding a second material over a first part.
Flash — Thin waste plastic escaping at the parting line.	**Insert moulding** — Moulding plastic around a pre-placed metal insert.