

Injection moulding



Polymer Injection Moulding- background

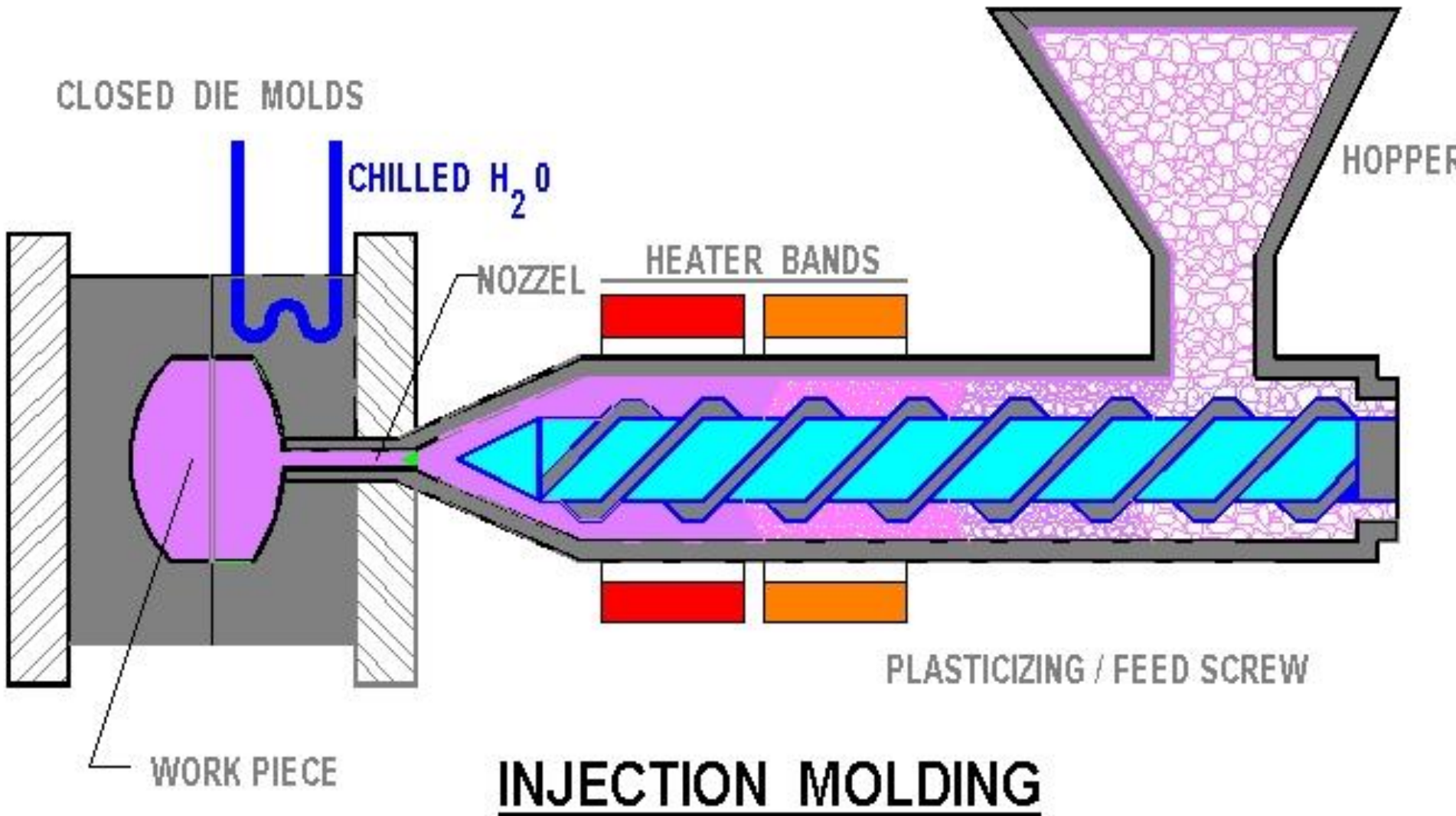
- The most widely used polymer processing method, around since 30's
- Initially ram injection with heated barrel, screw gives a better mix and temperature gradient
- Forming with molten polymers, thermoplastics and thermosets with high viscosity
- Large range of sizes and shapes, good for quite small items up to large items of a maximum 2-3m
- Highly automated
- Good control

Polymer Injection Moulding

- Hot melt extrusion and injection moulding are processes where the material is formed at high temperatures.
- Injection moulding shapes the material in 3D so can produce material in 1 processing step.
- Equipment- Hopper, screw mould
- Careful control of critical parameters; temperature, pressure, speed
- Compounds need to be stable at high temperatures to withstand without degradation
- Polymers-poor thermal conductivity-melt slowly

Advantages of injection moulding

- Reduction of process steps (integration of mixing, melting, forming and cooling)
- Highly consistent process due to holding period which allows for polymer shrinkage
- Low scrap weight
- Good for complex parts and high volumes
- Cost effective
- Good control of part size/shape – very automated



Additional Resources

[Injection moulding animation](#)

Six steps

- Mould closing
- Injection
- Holding
- Cooling
- Mould opening
- Ejection

- Continuous process
- Push faster = flows more easily, screw backs up and injects all at once. Volume at end of screw about 1.5 times mould cavity
- Need to match machine to mould size. If making large and small items could make several small at one go.
- Holding /packing maintaining pressure to allow for shrinkage
- Disproportionately expensive as you go bigger. More pressure needed for large parts and mould has to be ridiculously strong.
- Large items tend to be lower volume exceptions car bumper systems and sailing dinghies

Additional Resources

[Cap Injection](#)

Disadvantages

- Requires uniform wall thickness
- Can't do very large parts as a single piece
- Too expensive for one-offs
- Long lead time and high tooling costs
- Predominantly used with thermoplastics. When used with thermosets, the screw has to be cooler to prevent cross linking and the mould hotter
- Complicated computer modelling is required for the mould

Critical parameters

- Melt temperature- temperature of the molten polymer at the point of injection (minimised)
- Barrel Temperature (electrical heat bands)
- Cooling temperature (mould temperature) usually controlled by oil or water circulation.
- Holding/packing pressure (high enough to cope with polymer shrinkage on cooling)
- Holding/packing time long enough to fill mould cavity 100% without being long enough for backflow through gate
- Back pressure – pressure applied to material during screw recovery, aids mixing

- Injection Speed – injection moulding imposes high shear flow rates on the polymer as it is squirted at high pressure into the die aligning long polymeric molecules and increasing polymer fluidity. Shear proportional to injection speed.
- Cooling time set so mould solidifies quickly but polymers have long cooling duration and the time can effect crystallinity, physical stability etc

Nomenclature

- Sprue – entry point of injection moulding
- Runners – side channels
- Gates – thin part at end of sprue/runners where can break off
- Weld lines – last bits for resin to reach where doesn't mix properly. Weak point which can be improved by faster injection speed or hotter. Hotter means longer to cool thereby slowing production rate
- Daylight-amount mould opens
- Inserts used for undercuts to make a hollow item as opposed to solid
- Ejector pins push part off mould
- Vents-small holes big enough to allow air out but not big enough for polymer to extrude
- Cooling channels – water channels used to cool mould
- Aspect ratio flow length/width
- Sink mark or voids are dips in outside or bubbles where the middle shrinks after the outer solidifies this is why uniform wall thickness necessary
- Computer modelling packages from firms like moldflow model where weld lines will be pressure etc to get mould right first time

Resin data sheet

Design for Manufacturing Course 5: Injection Molding - DragonInnovation.com



Resin	Use	Yield (MN/m ²)	Modulus (MN/m ²)	Heat Deflect (1.8 MPa : C)	Cost (\$/kg)	Shrink (cm/cm)
ABS	Housings	39	1,800	86	\$2.09	0.004
PC	Lenses	65	2,400	128	\$2.81	0.007
POM	Gears	65	2,800	105	\$1.48	0.02
Polyamide	Strength	123	2,750	65	\$3.63	0.013
PP	Food	36	1,380	93*	\$1.76	0.015
Aluminum		276	68,900		\$1.75	
Steel		435	205,000		\$0.39	

<http://www.matweb.com/>

http://www.mim.com.cn/eng/training/price_trend.asp

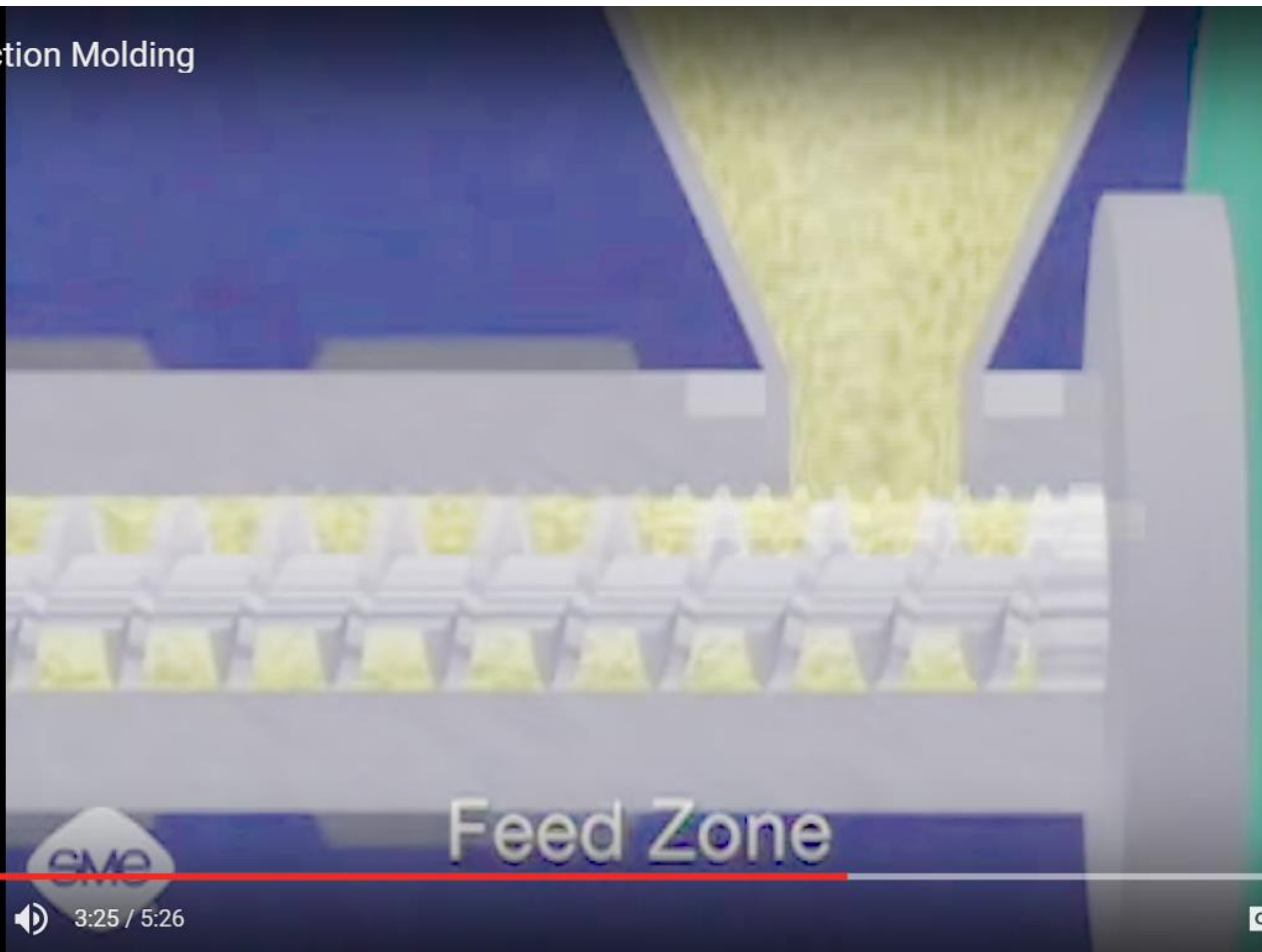
d



53:17 / 1:22:36



Plastic Injection Molding



Feed Zone



3:25 / 5:26



Plastic Injection Molding



Feed Zone



3:23 / 5:26





Melt Zone



3:33 / 5:26





Metering Zone



3:48 / 5:26

