INJECTION MOLDING PROCESSES

By

Muhammad Kevin Fahlevi

Supervised by

Dr. Kovács József Gábor

MŰEGYETEM 1782

Budapest University of Technology and Economics

Faculty of Mechanical Engineering

Department of Polymer Engineering

Budapest, November 2024

CONTENTS

1. Introduction

Injection molding is a highly adaptable process for manufacturing polymer products, particularly those made from thermoplastic materials. Small pellets of thermoplastic polymer are heated beyond their melting point, turning into a viscous liquid. This liquid is then rapidly and forcefully injected into a closed mold, which is maintained at a controlled temperature lower than the polymer's melting point. Within the mold, the pressurized polymer cools and solidifies into a precise, three-dimensional part, whether simple or intricate, practically without material waste.

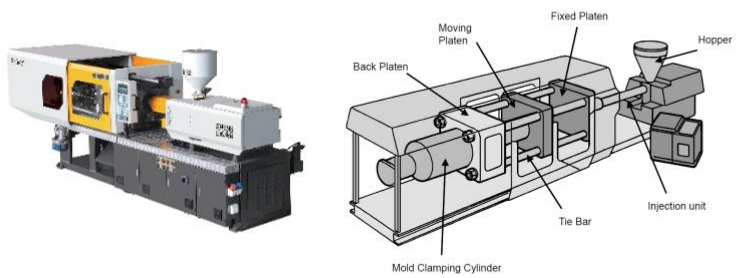


Figure 1. Injection Molding Machine

One notable characteristic of this technique is that a mold is designed for a specific product, making it unsuitable for creating different items. While the molds are expensive to produce, primarily due to the costs associated with machining the metal - each mold can generate a substantial volume of products, often exceeding 100,000 units. This high production capability results in a low cost per item, making the process cost-effective overall.

2. Injection Molding Machine

Injection molding is widely utilized for producing items of diverse sizes, shapes, and materials. To achieve efficient manufacturing, various types of injection molding machines have been designed and developed. Typically, an injection molding machine consists of several key components, as illustrated in figure below.

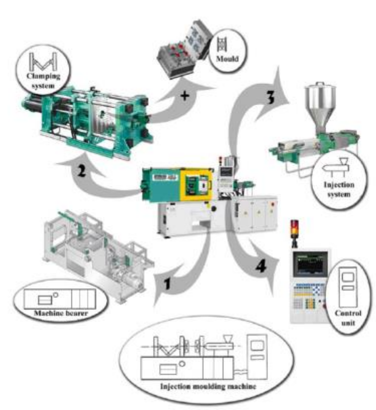


Figure 2. The main parts of the injection molding machine

The function of the machine stand (or bearer) and hydraulic system (Fig. 2 - 1) is to unite the other main systems of the machine and ensure a vibration-free connection between them. The function of the clamping system (or unit) (Fig. 2 - 2) is to move the movable mold half, while the stationary mold half is fixed to the mold platen. During the injection molding phase (when the cavity is filled with the thermoplastic melt), the mold must be closed against the high pressure of the molten polymer. The clamping force is one of the main parameters of an injection molding machine. When the part cools down to a certain temperature, the mold opens so that the product is removable. The type of mold clamps used in the injection molding machines can be direct hydraulic toggle clamp.

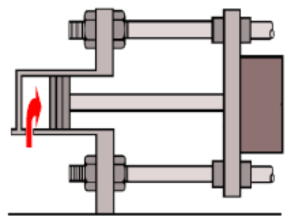


Figure 3. Direct hydraulic mold clamp

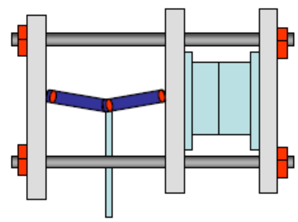


Figure 4: Toggle mold clamp

The injection system (Fig 2 - 3) is the most important unit in the injection molding machine. Its function is to melt the processed material (creating a homogenous melt from the thermoplastic granules) and inject it into the mold. The function of the control unit (Fig 2 - 4) is to control the main parameters of the injection molding machine and the whole production process and provide an interface between man and machine. A mold is also required for injection molding, but it is not listed as a main system of the injection molding machine since a mold can be used in several machines. Moreover, a machine can operate with various molds. The mold can typically be opened into two halves (a stationary and a movable mold half), which is necessary to remove the product from the mold cavity.

3. Injection Molding Material

Materials suitable for use in injection molding machines include:

- Thermoplastics; a type of polymer that becomes soft and moldable when heated and solidifies upon cooling. This process is reversible, meaning the material can be repeatedly melted and reshaped without significant chemical changes, making thermoplastics highly versatile and recyclable.
    - Semi-crystalline thermoplastics: Examples include LDPE, HDPE, LLDPE, PP, PA, POM, and PET.
    - Amorphous thermoplastics: Examples include PVC, PS, SAN, ABS, PMMA, and PC.

- Thermosetting polymers; or thermosets, are polymers that undergo an irreversible chemical reaction during the curing process, forming a rigid and cross-linked molecular structure. Once set, these materials cannot be melted or reshaped, even when exposed to heat. This makes them ideal for high-temperature and structural applications.
    - Elastomers (lightly cross-linked): Such as natural rubber, thermoplastic elastomers, and polyurethane.
    - Duromers (heavily cross-linked): Such as unsaturated polyester and epoxy resin.

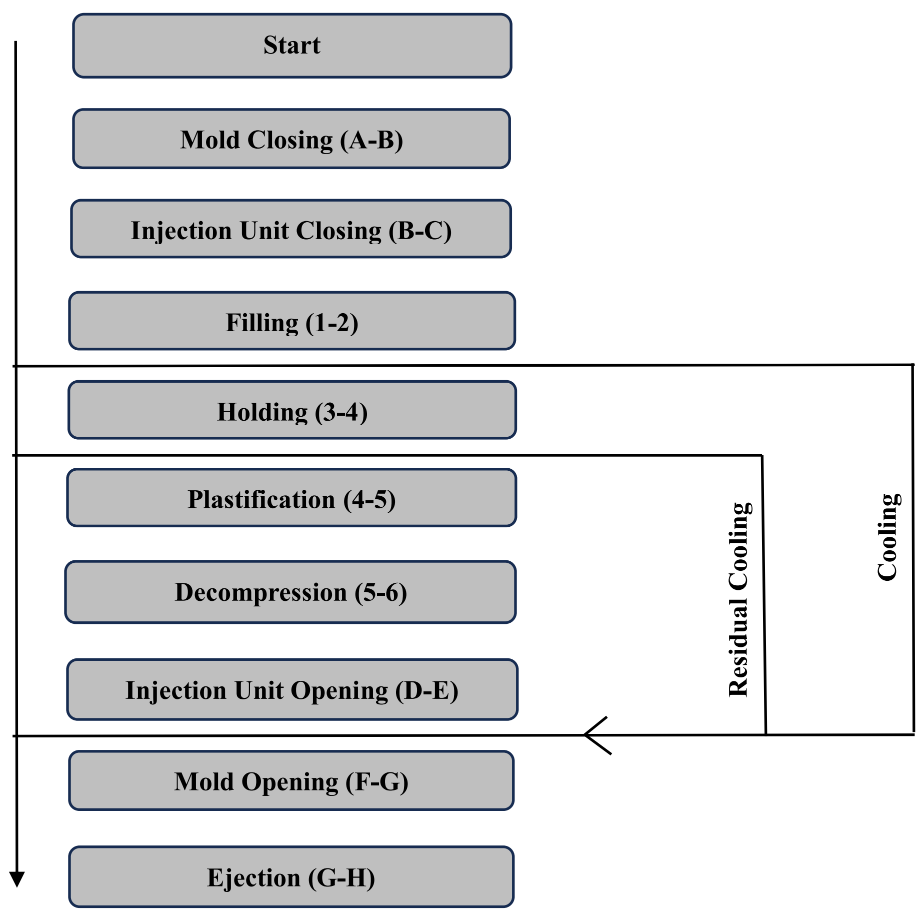
4. Injection Molding Processes

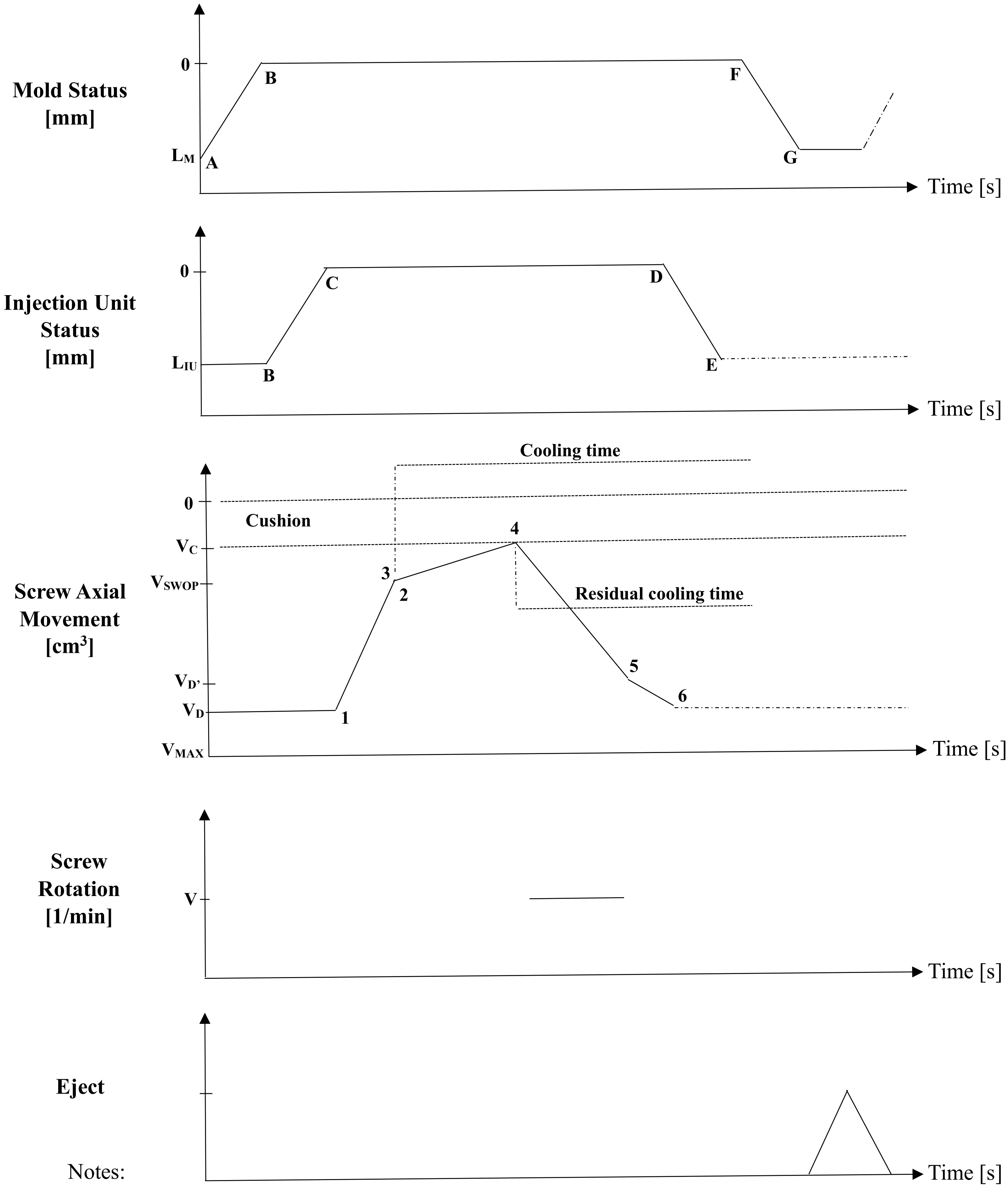
In general, the mode processes for the injection molding system are divided into two types of methods which are conventional and electrical. In our laboratory practice, conventional injection molding machine is used.

4.1 Conventional Injection Molding Stages

Conventional Injection molding is the method of injection molding which operates without simultaneous or parallel movements, utilizing an open nozzle design. It is characterized by sequential movements, where actions like mold opening and closing, injection, and part removal occur one after the other. This process is widely used for producing large quantities of uniform products efficiently.

The cycles of this method are shown below:




Mold Status [mm]
0
B
F
L_M
A
G
Time [s]
Injection Unit Status [mm]
0
C
D
L_{IU}
B
E
Time [s]
Cooling time
0
Cushion
4
V_C
3
V_{SWOP}
2
Residual cooling time
Screw Axial Movement [cm^3]
5
$V_{D'}$
6
V_D
1
V_{MAX}
Time [s]
Screw Rotation [1/min]
V
Time [s]
Eject
Notes:
Time [s]

- L_{M} : Length of mold
- L_{IU} : Length of injection unit
- V_{D} : Velocity of dosage
- V_{SWOP} : Velocity of switch over position
- V_{C} : Velocity of Cushion
- V_{MAX} : Maximum velocity

4.2 Data of the Equipment

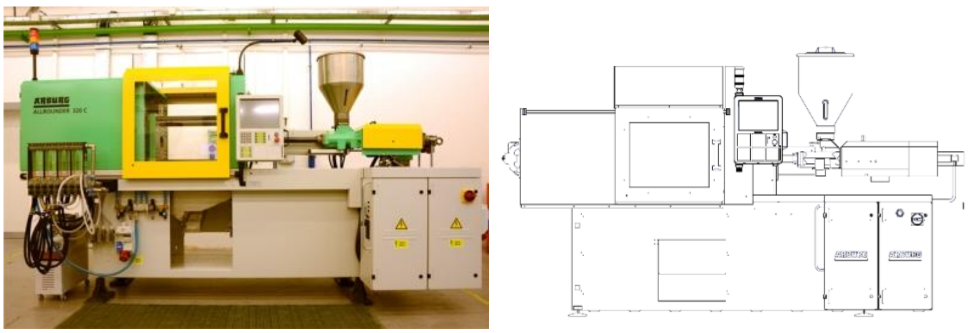


Figure 5. Direct hydraulic machine

Data of machine:

- Type: Arburg Allrounder 320C 400-170
- Manufacturer: ARBURG GmbH Max.
- Clamping system: Direct hydraulic
- Injection pressure: 2000 bar
- Screw diameter: 30 mm
- Max. clamping force: 40 t
- Max. shot weight: 88 g (PS)
- Distance between tie bars: 320 mm

Data of mold:

- Type: Tempro mold temperature controller series
- Manufacturer: WITTMANN BATTENFELD GmbH
- Temperature: 25 - 50 °C

Data of used material:

- Name: Polypropylene TATREN HM 50 46
- MFR (230 °C/2.16 kg): 50 g/10 min
- Tensile stress at yield: 37 MPa
- Tensile strain at yield: 8 %
- Modulus of elasticity in tension: 1900 MPa
- Flexural modulus: 1900 MPa

- Izod impact strength (notched, 23 °C): 3 kJ/m²
- HDT (0.45 MPa, flatwise): 100 °C
- Rockwell hardness: 107 R scale
- Processing temperature: 190 - 250 °C

Description:

TATREN HM 50 46 is a controlled rheology homopolymer of high fluidity and narrow molecular weight distribution. This grade contains nucleating and antistatic agent. TATREN HM 50 46 is characterized by very good mechanical properties, good dimensional stability and excellent stiffness. It has higher crystallization temperature which allows good de-molding at higher temperature and shorter molding cycles.

Application:

TATREN HM 50 46 is intended for thin wall injection molding of household and garden articles like containers, storage boxes, bowls, buckets, lids, trays, caps, boxes for food packaging, media boxes, flowerpots, garden furniture and other purpose articles. This grade can be used in mixtures with TATREN impact copolymers. Usage of TATREN HM 50 46 for compounding is possible as well.

4.3 Laboratory Processes

The stages of injection molding processes in our laboratory are consist of material selection, sequence, dry cycle, dosage, filling (SWOP), holding (gate freeze), optimization, and documentation. The details of each step will be described as follows.

- **Sequence (Full-Cycle)**

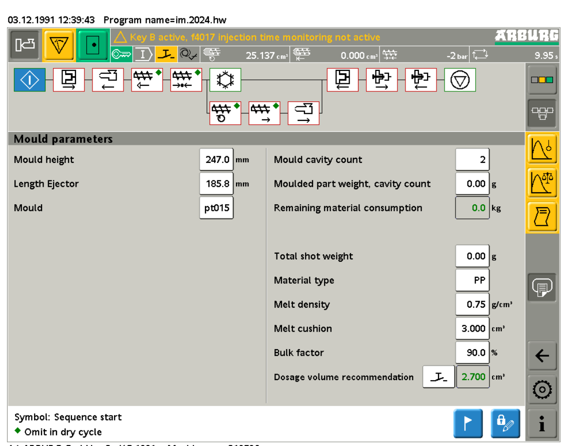


Figure 6. Mold parameter

The injection molding process begins with setting the key parameters for the molding cycle. This includes configuring mold height (247.0 mm), ejector length (185.8 mm), and material properties such as polypropylene (PP) with a melt density of 0.75 g/cm³. Dosage volume is calculated at 2.700 cm³ to ensure the appropriate material quantity is prepared for the molding cycle.

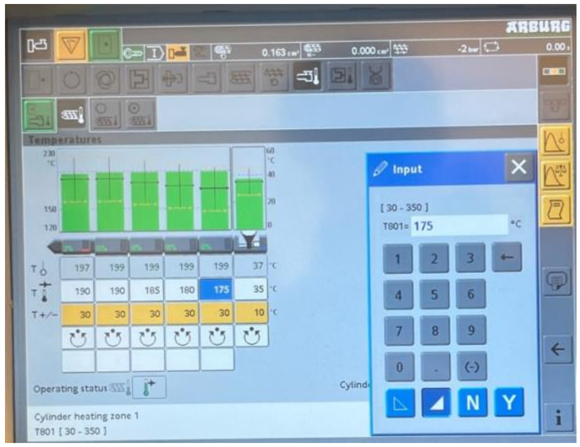


Figure 7. Cylinder temperature

In this step, the cylinder temperature parameters for the injection molding machine's heating zones are set to ensure the plastic material remains in a molten state during the injection process. The interface displays multiple zones with specific temperature settings, such as 197°C, 199°C, and 175°C, which are adjusted to maintain uniform heating and proper material flow. A precise input value of 175°C is entered for one of the zones, optimizing the melting of polypropylene to prevent degradation or insufficient melting. The machine indicates active heating, ensuring all zones are consistently monitored for reliable operation throughout the molding cycle.

➢ Dry Cycle

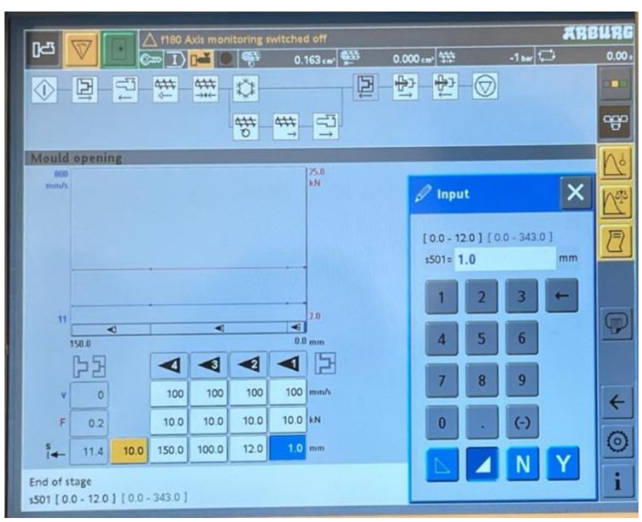


Figure 8. Mold opening

Before the mold closes, it’s needed configure the mold opening parameters, ensuring the mold halves separate smoothly after the injection process. The speed (800 mm/s) and force (2.0 kN) are precisely controlled to allow smooth separation of the mold halves after injection.

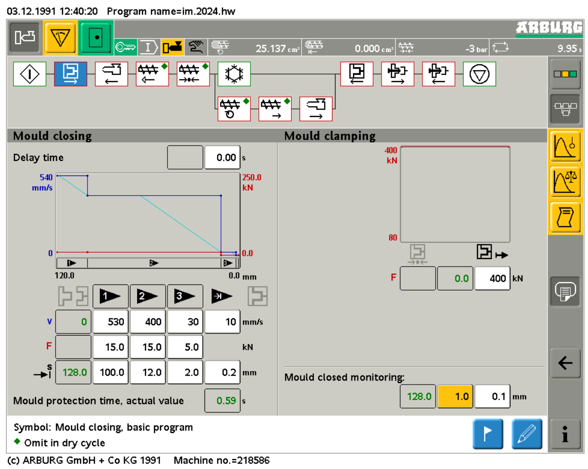


Figure 9. Mold closing

In this step, the mold closing and clamping phases are configured to ensure the mold operates safely and efficiently. The mold closing speed is initially set to 540 mm/s, gradually reducing to 0 mm/s as the mold fully closes, preventing damage to the tooling or machine. Once closed, the mold is clamped with a force of 400 kN, ensuring a tight seal during the injection process to avoid material leakage and maintain part precision. The mold protection time is set to 0.59 seconds, providing additional safety to avoid excessive stress on the mold. Monitoring parameters include a position setting of 128.0 mm, which ensures the machine confirms proper closure before proceeding with the next phase of the injection cycle.

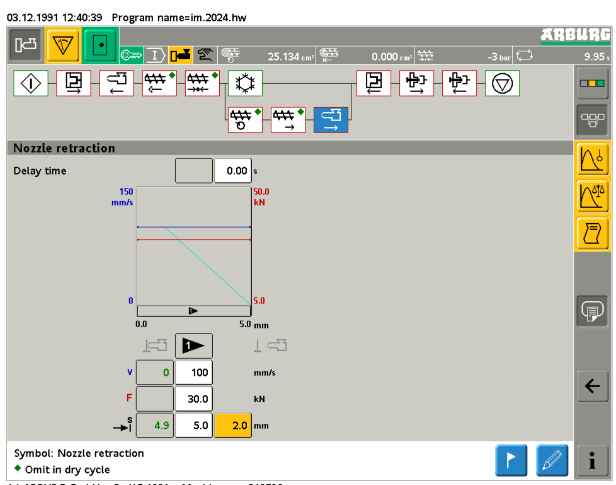


Figure 10. Nozzle retraction / Injection unit opening

In the nozzle retraction step, the nozzle is moved away from the mold after the injection phase to ensure it does not obstruct the mold opening or interfere with the ejection process. This retraction also prevents material leakage or stringing. The retraction speed is set to 150 mm/s, with a force limit of 50 kN, and the nozzle is retracted by 5.0 mm, monitored to ensure a position of 4.9 mm. There is no delay time applied, meaning the retraction occurs immediately after the injection phase is completed.

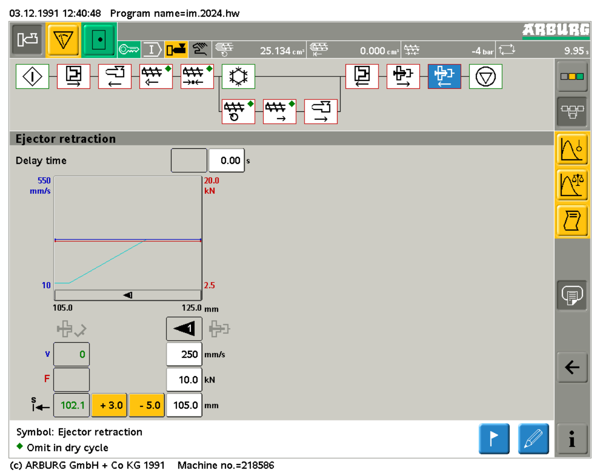


Figure 11. Ejector retraction

In the ejector retraction step, the ejector system retracts the pins to their original position after ejecting the molded part, resetting the machine for the next cycle. The retraction speed is set to 250 mm/s, with a maximum force of 10 kN to ensure smooth and efficient movement. The retraction distance is configured as 105.0 mm, ensuring the pins are fully withdrawn to avoid interference with the mold closing process. With no delay time applied, the retraction occurs immediately after ejection, and the position is precisely monitored, with a final position of 102.1 mm and tolerances of +3.0/-5.0 mm for accuracy and safety.

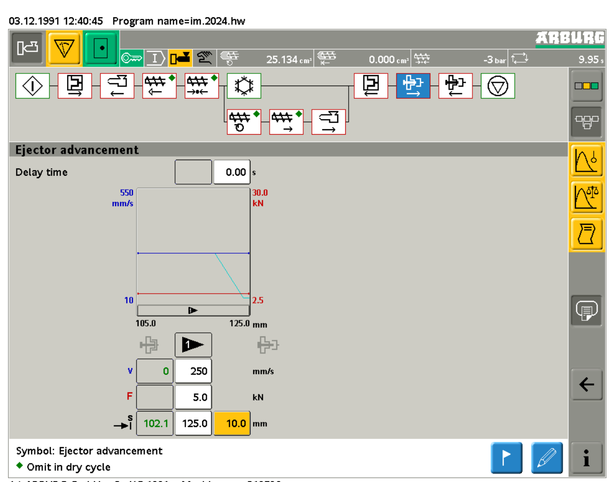


Figure 12. Ejector advancement

In the ejector advancement step, the ejector pins are pushed forward to eject the molded part from the mold cavity, ensuring it is released properly and without damage. The advancement speed is set to 250 mm/s, with a maximum force of 5.0 kN applied to move the pins efficiently while preventing excessive force that could damage the mold or the part. The advancement distance is configured as 10.0 mm, ensuring precise operation for clean ejection. To use the ejector, the mold must be open fully, otherwise the system won't allow the usage of ejector. Spring operated the ejector should always be in the back position.

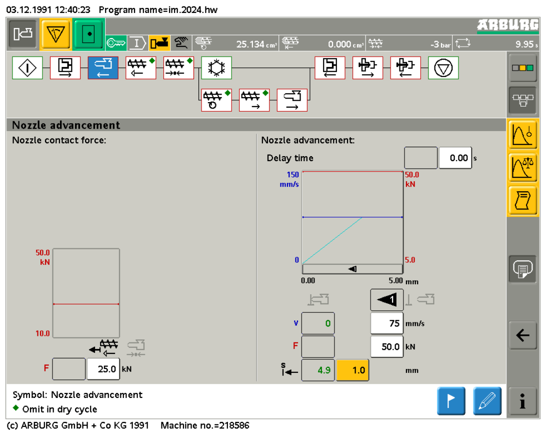


Figure 13. Nozzle advancement / Injection unit closing

In the nozzle advancement step, the nozzle moves forward into its operating position, aligning it with the sprue or gate to ensure accurate injection of the molten plastic into the mold cavity. The advancement speed is set to 150 mm/s, with a maximum force limit of 50 kN to guarantee smooth and precise movement. The nozzle travels 5.0 mm, monitored to ensure it reaches the correct position, verified at 4.9 mm. A nozzle contact force of 25.0 kN is applied to maintain a proper seal between the nozzle and the mold during the injection phase. No delay time is set, so the advancement occurs immediately when the step is initiated.

In this stage, the dry cycle mode should be turned on by turning on the automation logo. Those who are marked/highlighted on the screen will be extruded for the dry cycle. To avoid the problem, zero the mold can be applied by changing the temperature of the mold in regard to the heat expansion. After home safe, the injection unit and the mold shall be re-opened as these are the beginning of the cycle. In between two cycles, the injection unit must be always in the back position, and mold is always open.

➢ Dosage

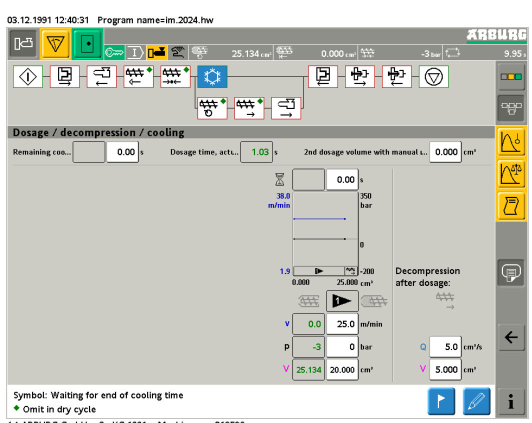


Figure 14. Dosage/decompression/cooling

The dosage phase involves the screw rotating to melt and accumulate a new volume of molten plastic, set to 20.00 cm³, at a speed of 25.0 m/min. Simultaneously, decompression occurs to relieve pressure in the molten plastic after the dosage phase, with a decompression volume of 5.00 cm³ at a flow rate of 5.0 cm³/s, ensuring material does not leak or string from the nozzle.

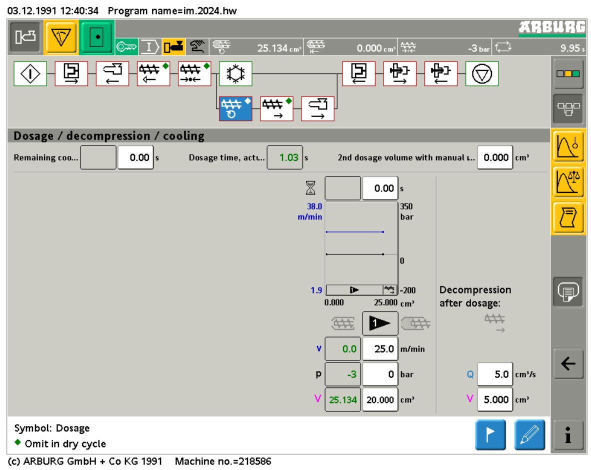


Figure 15. Dosage/decompression/cooling

The screw melts and accumulates 20.00 cm³ of molten plastic at a speed of 25.0 m/min for the next injection. Decompression is set to 5.00 cm³ to prevent nozzle leakage, while cooling ensures the molded part solidifies. The recorded dosage time is 1.03 seconds.

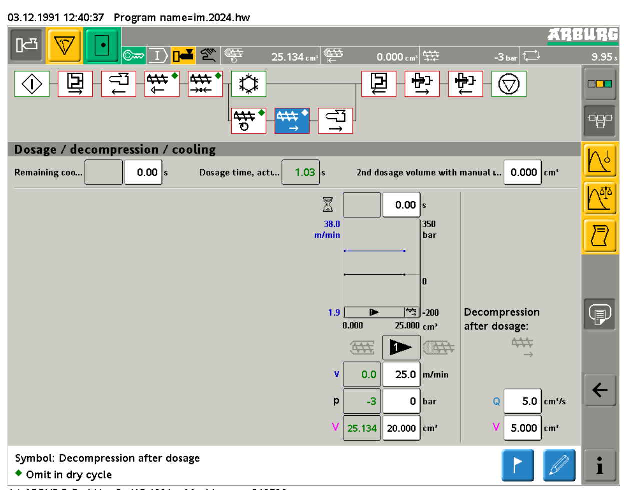


Figure 16. Dosage/decompression/cooling

This step emphasizes decompression after dosage, relieving pressure with a decompression volume of 5.00 cm³ at 5.0 cm³/s. Cooling continues to stabilize the molded part while preparing the system for the next cycle.

➢ Filling (SWOP)

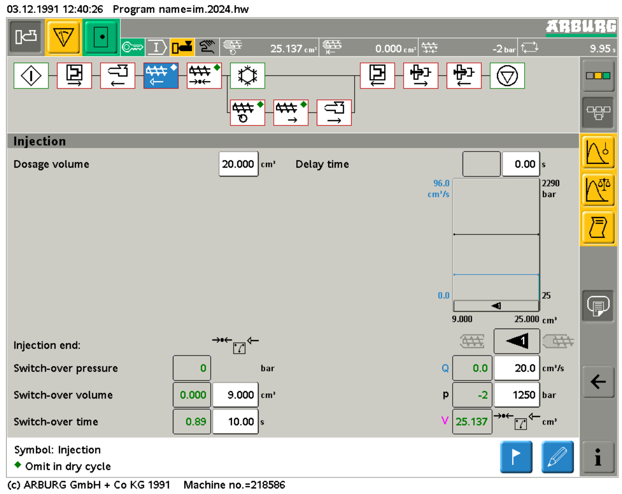


Figure 17. Injection

In the injection phase, the molten plastic is injected into the mold cavity to form the desired part. The dosage volume is set to 20.00 cm³, indicating the precise amount of material to be delivered into the mold. The injection speed is regulated to ensure consistent flow, with a maximum pressure of 2290 bar to fill the cavity efficiently while preventing defects such as voids or flash. A delay time of 0.00 seconds ensures the process begins immediately after the preceding phase.

The injection process includes a switch-over point, where the system transitions from high-speed filling to holding pressure. The switch-over is defined by a volume of 9.000 cm³, a pressure of 0 bar, and a time of 0.89 seconds to achieve optimal packing and minimize shrinkage.

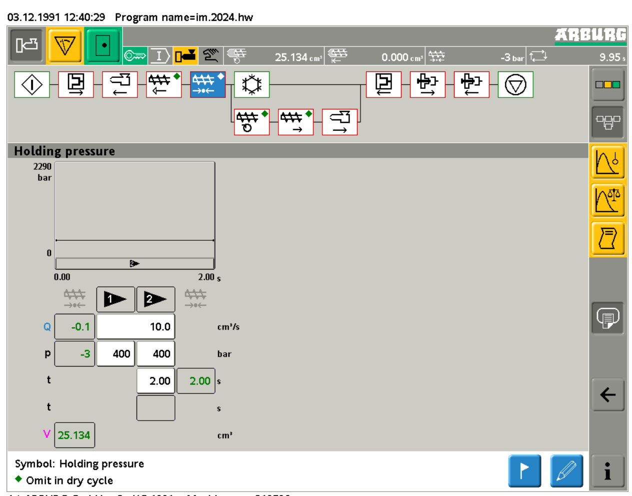


Figure 18. Holding pressure

In the holding pressure phase, pressure is applied to the molten plastic inside the mold cavity to compensate for shrinkage as the material begins to cool and solidify. The holding pressure is set to a maximum of 400 bar, ensuring that the cavity remains fully packed, and the molded part retains its dimensional accuracy. The volume of material held is 25.134 cm³, with the process lasting for 2.00 seconds to allow sufficient time for the plastic to stabilize within the mold. The flow rate during this phase is controlled at 10.0 cm³/s, maintaining consistent pressure distribution. In this phase, the additional time during cooling can be added to make sure the freezing of the gate. Two parameters can be checked and can be executed if the part is not fully complete: rise the pressure limit and reduce the switch-over-position.

Figure 19. Imperfect part

The incomplete part shown could result from the following issues:

1. The dosage volume may have been too low, leaving the cavity partially unfilled.
2. Inadequate pressure might have failed to push the molten material into all sections of the mold.
3. Premature cooling could have solidified the material before it filled the entire mold cavity.
4. A blockage in the gate or runner system may have restricted material flow.
5. The injection or holding phase may have ended too early, not allowing the cavity to fill completely.

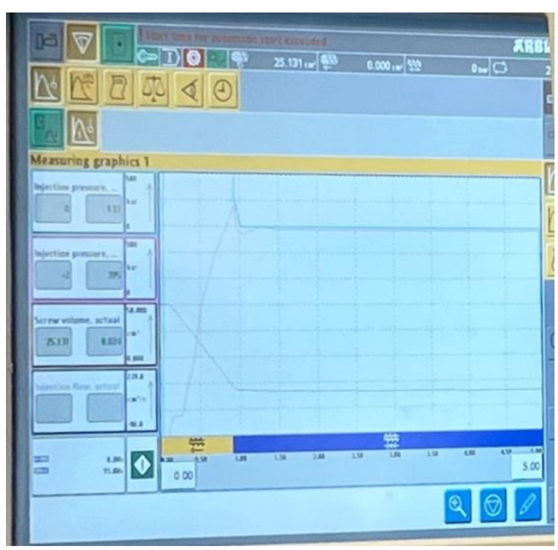


Figure 20. Incorrect measuring graphics

The graph shows an irregular spike and increase in injection pressure during the holding phase, where it should decrease smoothly. To fix this, the volumetric flow rate should be increased to 10 cm³/s during holding to stabilize the pressure.

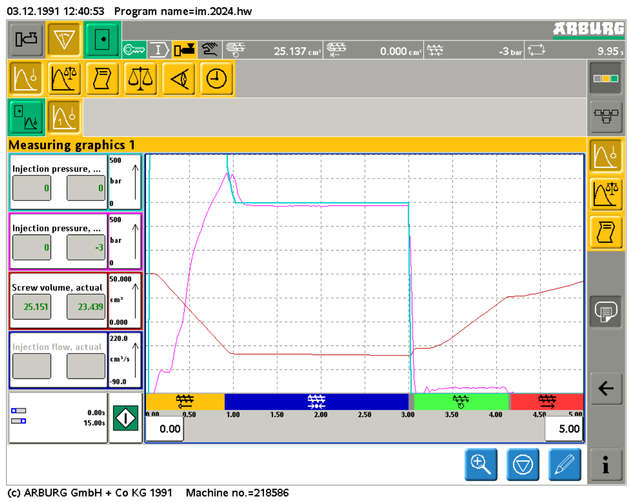


Figure 21. Correct measuring graphics

The graph shows a smooth injection process, with injection pressure peaking during filling and gradually decreasing during holding. The screw volume decreases steadily as material is injected, and the flow rate transitions smoothly.

➢ Holding (Gate freeze)

tH [s]	mn [g]		
0	7,17	7,176	7,181
1	7,47	7,477	7,491
1,50	7,59	7,573	7,579
2	7,646	7,666	7,656
2,5	7,688	7,656	7,672
3	7,642	7,667	7,661
3,5	7,691	7,686	7,687

Figure 22. Table of mass measurements with various holding times

Another critical parameter to optimize is the holding time. Increasing the holding time leads to an increase in the part's mass until it eventually stabilizes. The goal is to determine the minimum holding time required to achieve the maximum mass of the complete part. This time is influenced by the size of the gates; in our cases, the gates are small, so assumption that the optimal holding time will be around 3 seconds can be accepted. To test this, measurement of the part's mass can be done at various holding times, repeating each measurement three times to account for uncertainties and minimize errors.

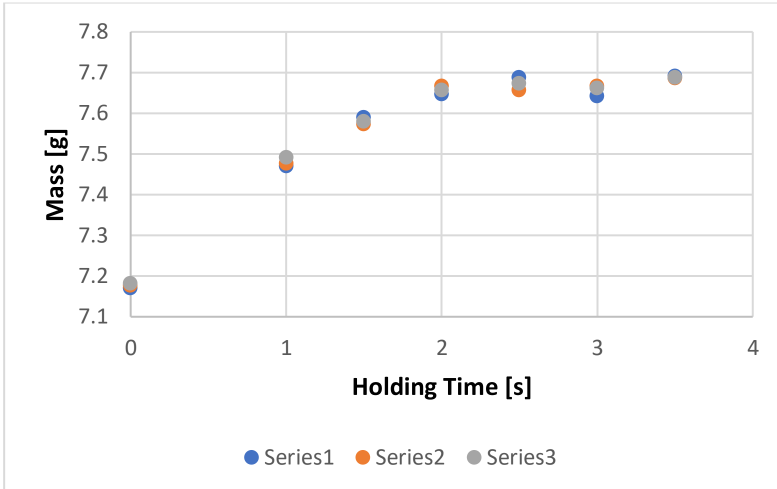


Figure 23. Graph of the mass and holding time

As shown in the figure above, the mass appears to stabilize after 2.0 seconds. However, without additional measurements, it is safer to consider 2.5 seconds as the optimal holding time as this is the middle point of the stabilized curve.

➢ Optimization

Figure 24. Optimized cycle time phases

The two images illustrate the cycle time diagram for the injection molding process, with a total cycle time of 9.95 seconds, which aligns with the optimization target of approximately 10 seconds. The breakdown of the steps for optimization are described as follows:

- Set all delay times to zero: Eliminate unnecessary delay times in all phases first, as this is a quick adjustment that directly impacts cycle time without affecting quality.
- Reduce cooling time: Evaluate whether the holding phase provides sufficient cooling, and if so, minimize or eliminate the cooling time.
- Reduce holding time: If part quality is unaffected, shorten the holding time to save additional cycle time.
- Increase mold movement speed**:** Accelerate the mold opening and closing phases while ensuring sufficient force is applied to maintain proper alignment and functionality.
- Minimize mold-to-nozzle Distance: Reduce the mold opening distance to the minimum required for part ejection, avoiding unnecessary travel.
- Increase ejector speed: Increase the speed of the ejector during both advancement and retraction phases to speed up part removal.
- Optimize nozzle advancement Speed: Increase the nozzle movement speed to minimize transition times while ensuring proper alignment with the mold.
- Keep screw speed constant: Do not alter the screw rotation speed to avoid affecting the melt temperature and introducing additional setup requirements.

5. Conclusion

The injection molding process involves several critical stages: setting up parameters, preparing the machine, and executing the production cycle. Key steps include nozzle advancement, injection of molten material, holding pressure to pack the mold, cooling, and ejecting the part. Each phase must be precisely timed and controlled to ensure consistent quality. Once the part is ejected, the cycle repeats with minimal downtime.

Optimization focuses on reducing cycle time without compromising quality. This includes eliminating unnecessary delays, minimizing cooling, and holding times, increasing mold and ejector speeds, and optimizing nozzle movements. Throughout the process, maintaining consistent screw speed ensures stable melt temperature and avoids the need for recalibration. By refining these steps, the process achieves greater efficiency, reduced costs, and consistent part quality.

REFERENCES

1. pt.bme.hu/
2. Injection moulding machine manufacturer | ARBURG
3. Tatren_HM_50_46_ENG_1807.pdf
4. https://www.polyplastics.com/en/support/mold/outline/
5. https://zetarmold.com/injection-mold-production/