



## Design and Cooling Performance of Plastic Injection Mold

Kemal Ermis

Sakarya University of Applied Sciences, Faculty of Technology, Mechanical Engineering, Sakarya, Turkey

Accepted 12 May 2024

### Abstract

The use of plastic products has increased considerably in the last century. Plastic products are used in many areas and places in our daily lives. Injection molding is the most widely used method in the production of plastic products. In this study, the mold design of a selected plastic part was made in a computer environment. The improvements required and made during the design are explained. While designing the plastic part, the draft angle was given to the surfaces required for the plastic part to come out of the mold comfortably. Since the plastic part will make some shrinkage when the plastic part is left to cool, the plastic part dimensions were increased by 5% by the standards while designing the mold design. Considering that the raw plastic material can reach the mold cavities most easily, the runner design was made. Analysis of the mold design was carried out. The calculations required for the cooling system were made and cooling channels were designed. At the end of the part production in the plastic mold, the cooling process was analyzed using the package computer program and the results were examined.

**Keywords:** *Plastic injection, Cooling analysis, Mold design, Molding*

### 1. Introduction

The use and production of plastic products has been increasing rapidly in recent years. Since plastic materials and manufacturing are cheap and easy, production quantities are increasing. Injection molding is a popular method that is used to make plastic products due to high productivity, efficiency and manufacturability [1]. Today, 32 percent of the plastic-based products produced come from injection molding machines. It is one of the most common methods used for the mass production of complex plastic parts. One of the most important processes in the production of plastic materials is the part mold design and cooling process [2].

Plastic injection mold design and the design of the mold cooling system affect the quality, surface roughness, geometric dimensions of the product. This also affects the mold cycle time and cost. The basic principle of injection molding is, that hot plastic is to send it into the mold with pressure and wait for the melt to cool and harden by taking the shape of the mold cavity. The filling and packing stage, the cooling stage, and the ejection stage are the three main steps in the injection molding process. The cooling step is the most crucial of them since it primarily influences

molding quality and productivity [4-7]. In the first phase of the development process, the research results show that conformal cooling channels reduce the cycle time by approximately 30% compared to conventional cooling channels [1]. Mold temperature is important as it determines the printing time.

Although molten plastic flows easily in a hot mold, it takes a long time for the product to cool down to the temperature required to be removed from the mold after cooling. The design of the cooling system is therefore important. Cooling too fast causes the molten plastic in the mold to cool and harden before filling the mold. If the cooling system is not designed in accordance with the produced part, defects such as shrinkage, distortion and collapse may occur in the part [8-10]. When designing cooling systems, the part should be analyzed and the most suitable cooling system should be selected [11-13].

In this work, a computer environment was used to create the mold design for a particular plastic part. The design explains the necessary and achieved improvements. The surfaces needed for the plastic part to comfortably exit the mold were given the draft angle during the plastic part's design. The dimensions

of the plastic part were increased by 5% in compliance with requirements during the mold design process because the plastic part will shrink somewhat when it is allowed to cool. The runner design was created with the concept that the raw plastic material may enter the mold spaces most straightforwardly. The design of the mold was analyzed. Cooling channels were developed and the necessary computations for the cooling system were performed. Using the computer application included in the box, the cooling process was checked after the item was produced in the plastic mold. The cooling system was also analyzed and interpreted in the CREO program.

## 2. Plastic injection mold design

The plastic injection molding process involves a continuous cyclical process. This process is completed in four important stages. These stages are filling, packaging, cooling, and ejection. The plastic injection molding process begins when the resin and appropriate additives are fed into the injection molding machine from the hopper into the heating and injection system. The filling stage is where the mold cavity is filled with the polymer melt at injection temperature. Once the cavity is completely filled, in the packaging stage, additional material melt is packed into the cavity at a higher pressure to compensate for the expected shrinkage as the plastic melt solidifies. Once the cavity is completely filled, in the packaging stage, additional material melt is packed into the cavity at a higher pressure to compensate for the expected shrinkage as the plastic melt solidifies. This is followed by the cooling stage, where the mold is cooled until the part is hard enough to be ejected. The final step is ejection, where the mold is opened and the part ejected, after which the mold is closed again to start the next cycle.

The plastic mold processing process is very delicate and complex, a sequence of operations involves many steps that depend on each other, and each step requires strict work and control. Plastic mold design is the first step of the whole processing process, and at this stage, it needs to use CAD software to realize the three-dimensional design of the mold according to the specific mold structure, size, structure and functional requirements of the product. To ensure that the plastic can fill every part of the mold evenly, simulation of the plastic filling process and flow channel analysis must be performed for error-free and high-quality production. The accuracy and rationality of the design stage will directly affect the quality of subsequent processing and molding. The second stage is material selection. Material selection is an important part of plastic mold processing. It is necessary to select the

appropriate mold steel according to the use of the mold, service life, type of plastic and other factors. In general, the mold steel must have high hardness, high wear resistance, high strength, high corrosion resistance and other desired specific properties. Choosing the wrong material will cause the mold to have a short life, easy wear, easy deformation and other problems, which will seriously affect production efficiency and product quality.

In order to produce a plastic part by injection molding, there are many issues to be considered in the design of the part, material selection and production. First and foremost, the product design must be functional and fulfill its purpose. When designing the product, tie pieces should be placed in the necessary places, the production stages should be carried out in the shortest possible time and avoid unnecessary weight. Tie pieces are structural backbones used in plastic parts to increase the strength of the part without affecting the functional areas and appearance of the part. Plastic parts may undergo deformation as a result of stresses occurring at the place of use. Tie-piece structures can minimize these deformations. Apart from these, things to be considered when designing an injection mold:

- Wall thickness
- Drop gate, runner head
- Draft angle
- Shrinkage rate

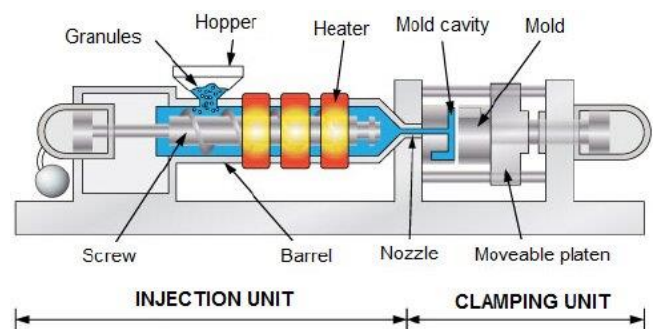


Figure 1. A Simple schematic of an injection molding machine [13].

In this study, PLA was chosen as the material to be produced for the mold design. Since PLA is bio-based, it turns into natural materials within 3 months when suitable conditions are provided compared to other plastics. PLA is a thermoplastic, meaning it can be melted and reshaped without losing its mechanical properties. A simple schematic of an injection molding machine is shown in Figure 1. There are two types of runner systems in injection molds: hot and cold runners. Hot runners are more costly and difficult to maintain and install. There is also a high probability of malfunction. Cold runners are more advantageous

than hot runners, are simpler to install and have lower costs.

The cold runner was chosen because it was cheaper and easier to install. With the circular cross-section, advantages such as low heat and friction losses and final freezing of the center are provided. With the successful implementation of the cooling system, positive improvements (quality, stability, cost) will be seen on the part. This method is preferred because it is suitable for mass production, has low material loss, has good mechanical properties, and is suitable for automation. In this study, the cold runner type was chosen in the mold design.

In mold design, some requirements must be met, such as determining the distance between the mold surface and cooling channels, determining the inlet temperature and flow rate of the cooling liquid, the density of the selected material, the temperature of the mold, the diameter of the cooling channel and the method by which the channels will be placed according to the desired features, and runner design. The product in this study is a simple Lego piece that was selected as shown in Figure 2. that can be produced more than once at a time. The technical drawing is given in the appendices. In the design of the product, attention was paid to factors that would facilitate mold design. Care was taken to use equal thicknesses everywhere instead of using different thicknesses to give a draft angle to the inner surfaces and to ensure that every part of the product cools in the same amount of time.

In the design of the product, attention was paid to factors that would facilitate mold design. Care was taken to use equal thicknesses everywhere instead of using different thicknesses to give a draft angle to the inner surfaces and to ensure that every part of the product cools in the same amount of time. Thanks to the given draft angle, the inner surfaces will rub less against each other when the product leaves the mold, and the product will come out of the mold easily.

The Mold design interface of the CREO program was used for mold design. First, the Lego part was designed in a solid model environment. After the design was completed, the mold design started by switching to the CREO mold interface. To make the mold design, the part designed was called and taken into account factors such as layout, number, casting direction and gaps between the products. The model produced with the package program is shown in Figure 3 and mold design is shown in Figure 4. The

nozzle inlet is made in the middle of the mold to distribute the loads evenly to the mold.

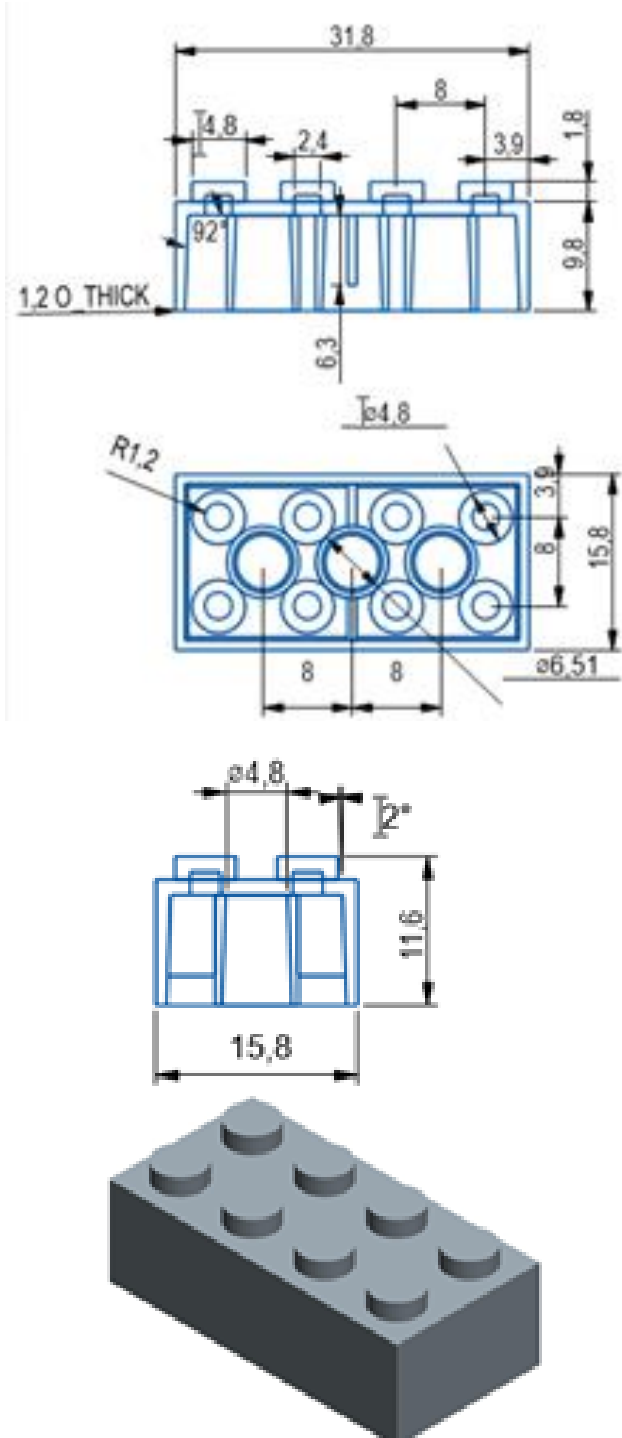


Figure 2. Dimensions of the Lego piece

Plates of desired size and thickness can be selected from the manufacturer's catalog. A mold is selected in dimensions that will fit the previously designed mold. Figure 5 shows fixing bolts, nozzle inlet, and pushers placed in the mold.

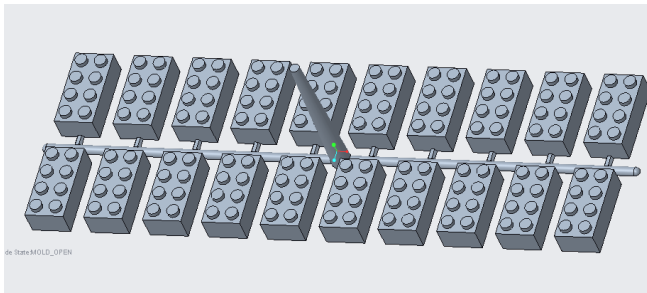


Figure 3. The model produced

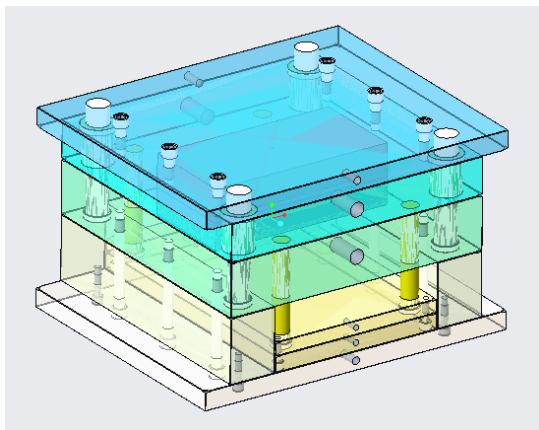


Figure 4. Mold design

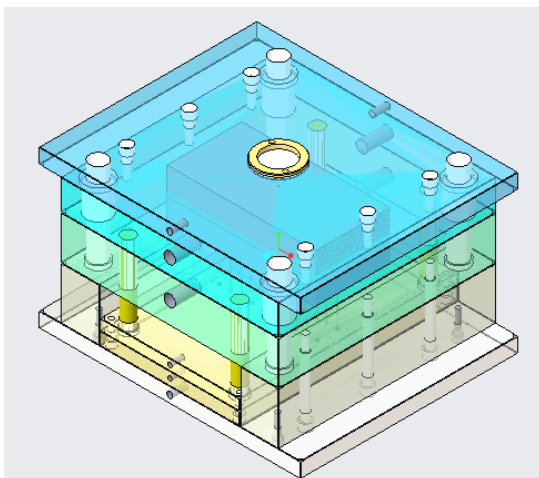


Figure 5. Fixing bolts, nozzle inlet, and pushers placed in the mold

The template was designed in the EMX module of the CREO package program. Parts are produced in a CAD environment according to the designed mold. The produced part is opened in the mold analysis module of the CREO program. First, the properties of the PLA material to be printed on the mold were defined. Then, the gate and diameter required for the fluid to enter are defined. After these operations are completed, analyses are carried out in the analysis section.

The mold ability feature of the model should first be analyzed. As shown in Figure 6, the green color indicates excellent mold ability.

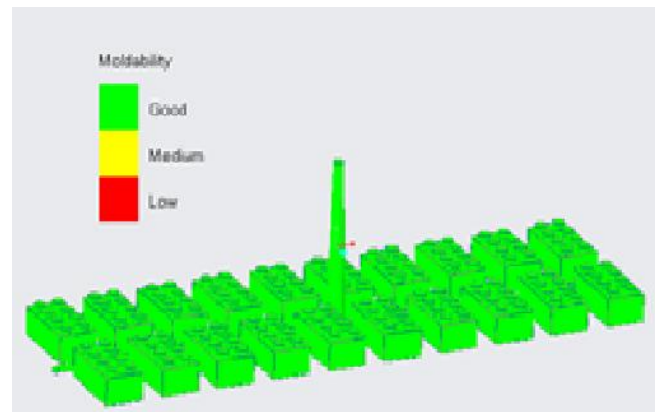


Figure 6. Mold ability analysis of the model

The collapse analysis can be seen in Figure 7. As seen in the figure, the sign of collapse is zero in the mold's eyes. It is only on the main road. Since this situation does not affect production quality, no changes will be made.

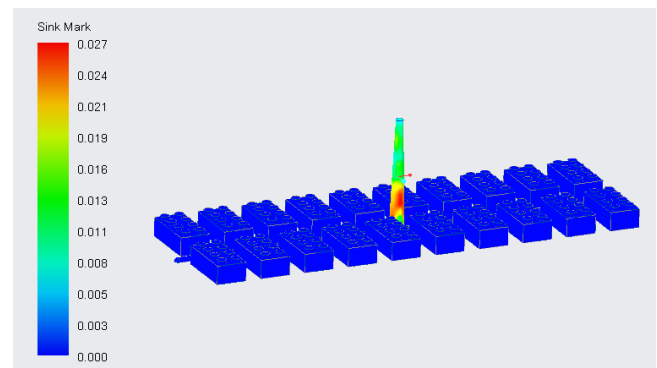


Figure 7. The collapse analysis

### 3. Cooling performance of plastic injection mold

The heat is given to keep the raw material in a molten state while filling the mold is taken back during the solidification process after the mold filling is completed. Cooling time constitutes the longest part of the cycle, a cooling system is needed to shorten the long process time. Heat transfer between the mold and the plastic part to be formed is another factor that affects the cost. Therefore, a successful cooling system is required. With this system, a positive contribution to the cost, quality of the surface of the part and decreases in the shrinkage rate can be observed. In mold design, the arrangement and number of parts, the placement of the runner system, and the homogeneous temperature distribution on the mold walls must be achieved for the plastic part to be produced to have a homogeneous surface structure. This method is used due to the ease of use of the mold and the rapidity that will occur in production. Successful temperature control must be provided to increase part surface quality, reduce unit costs and

improve mechanical properties. The main purpose of cooling water is to cool the part evenly.

The difference between the inlet and outlet temperatures of the cooling water is not desired to be more than 5-6 °C. Otherwise, the less solidified area will continue to cool and shrink after leaving the mold, and distortion will occur towards the less cooled side. Cooling water flow is an important parameter in the cooling system. A laminar flow is not preferred in the molding industry. This is because only certain particles participate in cooling in laminar flow. In turbulent flow, many particles affect the cooling channel walls. In this study, turbulent flux and Reynolds number were taken as 10000. The cooling channel is shown in Figure 8. The temperature distribution of the plastic melt at the end of filling the mold is shown in Figure 9.

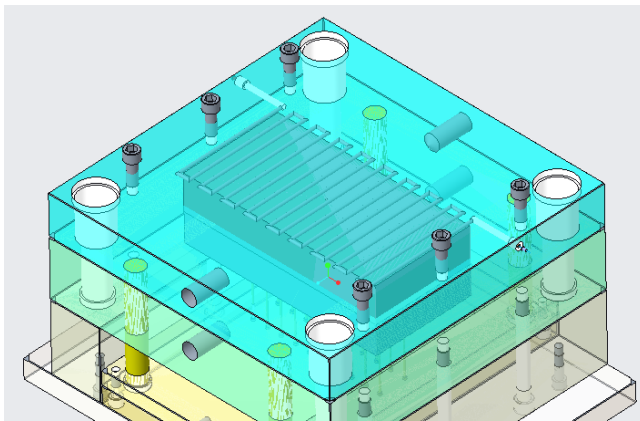


Figure 8. Cooling channel

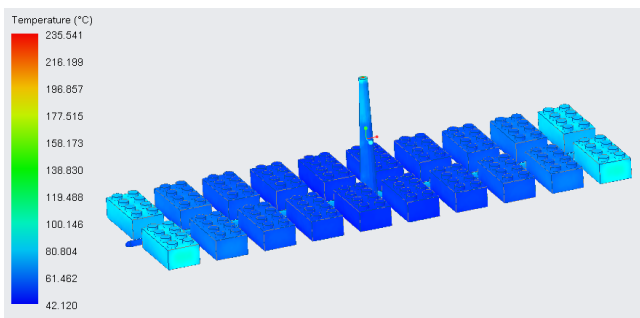


Figure 9. The temperature distribution analysis of the plastic melt at the end of filling the mold

The core temperature is the temperature of the middle layer in the thickness direction at the end of the filling. The core temperature is indicative of the thermal energy delivered to the new hot melt. If the core temperature is too low, flow stalling occurs, causing the molten material not to completely filling the mold's voids. The core temperature analysis is shown in Figure 10.

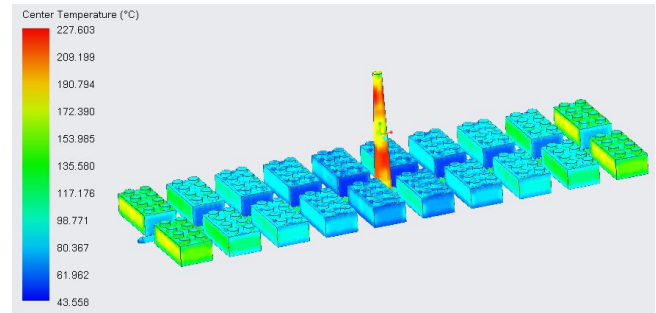


Figure 10. The core temperature analysis

The stack temperature is the velocity-weighted average temperature of the plastic melt through the thickness at the end of the filling. The contribution of the stationary frozen layer is neglected. The effect of heat convection and viscous heating can be observed from these data. The bulk temperature distribution reflects the trend of the flow path. The stack temperature analysis is shown in Figure 11.

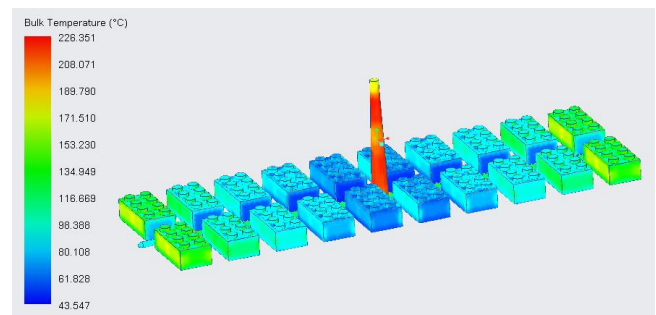


Figure 11. The stack temperature analysis

#### 4. Conclusions

In this study, the design of a plastic injection mold for the production of Lego from PLA material and the analysis of the temperature distribution resulting from the cooling process, which is important during this design, was carried out with the help of the CREO package program. The most important variable when designing the mold was the runner design and the placement of the parts in the mold. When you change the position of the parts in the mold or the shape and distance of the runners, the path of the melt in the mold will also change. Therefore, experiments were carried out according to various criteria until the best mold design was found and the best model was obtained. The necessary process for mold modeling and cooling systems has been covered. As a result, a successful mold design was achieved by establishing the criteria for mold design. Temperature distribution, core temperature and stank temperature analyses were performed in the cooling system and the results were shown.

### Acknowledgment

The author thanks Mertcan Yener (Mechanical Eng.) and Sefa Kızılcık (Mechanical Eng.) for helping with the analysis.

### References

- [1] Park H.S., Dang X.P. Development of a Smart Plastic Injection Mold with Conformal Cooling Channels, *Procedia Manufacturing*, 2017; 10: 48-59.
- [2] Park S.J., Kwon T.H. Optimal cooling system design for the injection molding process. *Polymer Engineering & Science*, 1998 38(9): 1450-1462.
- [3] Gürün H., Özdemir A, Acar T. Computer Aided Design of The Cooling System for Plastic Injection Molds, *Pamukkale University Journal of Engineering Sciences*, 2009;15:263–268.
- [4] Li C.G., Li C.L. Plastic injection mould cooling system design by the configuration space method. *Computer-Aided Design*, 2008; 40(3): 334-349.
- [5] Qiao, H. A systematic computer-aided approach to cooling system optimal design in plastic injection molding. *International Journal of Mechanical Sciences*, 2006; 48(4): 430-439.
- [6] Hassan, H., Regnier, N., Le Bot, C., Defaye, G. 3D study of cooling system effect on the heat transfer during polymer injection molding. *International Journal of Thermal Sciences*, 2010; 49(1), 161-169.
- [7] Hassan, H., Regnier, N., Lebot, C., Pujos, C., & Defaye, G. (). Effect of cooling system on the polymer temperature and solidification during injection molding. *Applied Thermal Engineering*, 2009; 29(8-9), 1786-1791.
- [8] Liang, J. Z. An optimal design of cooling system for injection mold. *Polymer-Plastics Technology and Engineering*, 2002; 41(2), 261-271.
- [9] Park, H.S., Dang, X.P. Design and simulation-based optimization of cooling channels for plastic injection mold. *New technologies: trends, innovations and research*, 2012; 19-44.
- [10] Gruber, P. A., & de Miranda, D. A.). Heat transfer simulation for decision making in plastic injection mold design. *Polimeros*, 2020; 30, e2020005.
- [11] Tang, S.H., Kong, Y.M., Sapuan, S.M., Samin, R., & Sulaiman, S. (). Design and thermal analysis of plastic injection mould. *Journal of materials processing technology*, 2006;171(2): 259-267.
- [12] Lin Z-C., Chou, M-H. Design of the cooling channels in nonrectangular plastic flat injection mold, *Journal of Manufacturing Systems*, 2002; 21(3): 167-186.
- [13] Svečko, R., Kusić, D., Kek, T., Sarjaš, A., Hančič, A., Grum, J. Acoustic Emission Detection of Macro-Cracks on Engraving Tool Steel Inserts during the Injection Molding Cycle Using PZT Sensors. *Sensors (Basel, Switzerland)*, 2013; 13: 6365-6379.