Understanding the Virtual Injection Molding Product Design

Alberto García-Collado and Rubén Dorado-Vicente

1 Introduction

During the design of a plastic component, the engineer considers both the technical requirements that the component will face during its service life, as well as the mechanical properties of the thermoplastic and the technology used in the manufacturing process. When launching new products, engineers must follow a series of stages, which are highly interrelated, such as the design process of the plastic component to be manufactured and the design of the mold where parts are to be injected.

As injected thermoplastic parts have gained importance in the field of engineering, the requirements they have faced have been increasingly demanding, not only in the design phase but also in the injection process. The distribution of the material inside the mold, as well as the flow orientations, and the different degrees of shrinkage that may result, must be calculated, and if necessary estimated before the manufacturing of the mold. All of this results in the great complexity of the design of injected plastic parts, since any variation in the mold will not only lead to a modification of the component design in a localized area, but it can also affect the rest of the part, reducing reliability and diminishing the mechanical performance.

Today, the Finite Element Method (FEM) is regarded as an indispensable tool during the design phase of both the plastic component and the injection mold. In the sense of engineering education, it is possible to develop methodologies based on training students' skills in commercial FEM software and at the same time use the software to explore complex engineering concepts by means of its powerful simulations. The present chapter describes this type of approach: the use of SolidWorks

A. García-Collado (✉) · R. Dorado-Vicente
Dpto. Ingeniería Mecánica y Minera, Universidad de Jaén, Jaén, Spain
e-mail: acollado@ujaen.es

R. Dorado-Vicente
e-mail: rdorado@ujaen.es

D. Carou and J. P. Davim (eds.), *Notes for Manufacturing Instructors*, Materials Forming, Machining and Tribology, https://doi.org/10.1007/978-3-031-48468-1_3

plastic as a tool to understand injection molding processes, and for training students in computer-aided tools to obtain part designs adequate for a manufacturing process.

2 Background

This section is devoted to summarizing the basic concepts regarding injection molding and FEM simulations required to understand the laboratory session. Roughly speaking, an injection molding process is based on the material phase change: a polymer is heated to its melting point and then introduced into a mold till it is filled. After that, the material solidifies with its final shape.

Figure 1 shows the main parts of an injection machine including the mold. The thermoplastic injection process can vary between 2 s and several minutes, depending on the type of part. The stages in which the process is divided are:

- Filling: the plastic is injected at high pressure into the mold.
- Packing: after injection, the material continues to be added to avoid a lack of material or density.
- Cooling: the material is cooled until a significant part of its volume solidifies.
- Extraction: opening of the mold and extraction of the piece.

In this lecture, SolidWorks® Plastics, a FEM application, to perform a simulation of the mold injection process till the packing stage is used. Generally, all injection molding simulation programs have the same internal structure:

- A preprocessor for geometry modeling.
- Databases that store the characteristics necessary for the analysis. Thus, a constant update of the database of commercial polymers, refrigerants, mold materials, and the characteristics of existing injection machines on the market is necessary.
- A series of specific calculation modules for each transformation process.
- A post-processor that friendly presents the results of the analysis.

SolidWorks allows to predict and avoid manufacturing defects in plastic part designs and injection molds, which eliminates costly modifications, improves part quality, and reduces time to market. In addition, it allows to balance channel systems and calculate cycle duration, closing tonnage and injection weight, which allows to optimize the design of the feeding system and avoid the high cost of having to re-make the molds.

Once the mold filling study has been carried out, the solution allows: knowing the best location and the number of material inputs in the mold if there are several inputs, a correctly balanced system will obtain the injection speed profile which minimizes the residual stresses of the component, the minimum mold filling time, evaluation of the welding lines or joint lines and air trapping in certain areas, determining the injection pressure and clamping force for correct sizing of the injection machine and the mold. All this information feeds back into the design phase since it can force a substantial

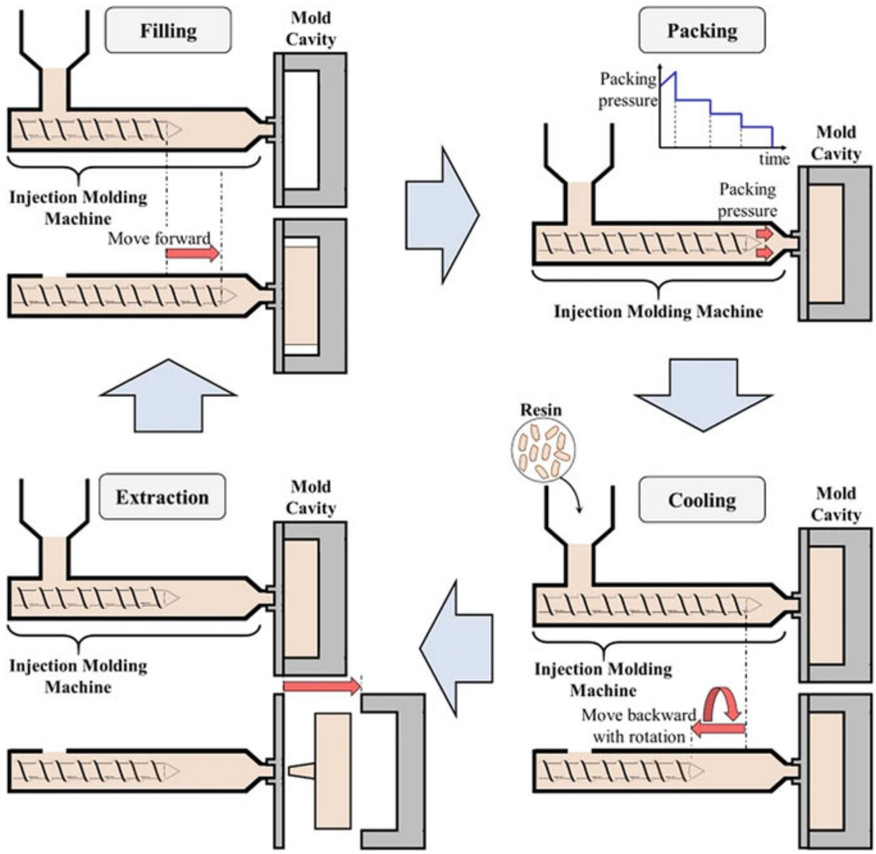

Fig. 1 Injection molding process stages. With permission of Springer. *Source* Jung, J. et. al. Optimization of injection molding process using multi-objective Bayesian optimization and constrained generative inverse design networks. https://doi.org/10.1007/s10845-022-02018-8

modification of the original idea of the component. Thermoplastic injection is the plastic molding technique consisting of injecting a molten polymer at high pressure through a nozzle (transferred by a piston mechanism or extruder screw), to fill a mold.

3 Learning Objectives

The following bullets summarize the learning objectives:

- Understand the main phases of the injection molding process.
- Get familiar with software for the analysis of the process.

- Know the process required for the simulation.
- Analyze an injection process determining the relationships between times, temperatures, and pressures.

4 Resources and Organization

To develop the required skills to attain the learning objectives (Sect. 3), the subsequent activities are proposed:

- Face-to-face lecture: a guided example with the commercial software based on the method of finite elements to calculate the feasibility of mold filling in plastic parts. An evaluation of the results of the simulated process, discussing aspects related to the design of the part, always oriented towards manufacturing is also accomplished.
- Homework: an exercise is proposed where students must perform a filling analysis like the guided example.
- Online test of different types of questions to reinforce the concepts taught during the face-to-face activity. This set is implemented in a Moodle-type educational platform.

The first activity is the face-to-face lecture. It is a 2 h session in a room with computers and the FEM software to use. The session development is described in the following Sect. 5.

During the ten last minutes of the face-to-face activity, the instructor explains the homework exercise and how to use the online test to face this work. Students will work in the exercise for one week, and in parallel, with the online test. This test is open one week before the due date for the homework activity. For every attempt, the responses are automatically evaluated, and the feedback is sent to the student.

The lecture is composed of two main activities as shown following:

Activity 1: Feasibility of injection mold evaluation example. Guided by the professor, the students review critical concepts and solve the mold evaluation example described in Sect. 5:

- The basic concepts of plastic injection will be briefly reviewed.
- The student will start working with SolidWorks® Plastics and its basic functions will be described by the instructor at the same time.
- Feasibility of injection analysis will be carried out with the software as an example.

Activity 2: Homework

- A part will be provided by the professor and discussed during the session.
- The part will be imported into SolidWorks plastic simulation, and the filling and packing process will be analyzed. The data collected will be used to complete a report according to the indications found at the end of this tutorial.

5 Session Development

This section is devoted to illustrating a laboratory session: face-to-face activity (Sect. 4). Through the session, it is described a feasibility analysis of injection plastic using SolidWorks® Plastics. Finite element simulation software is essential for designing plastic parts as it helps to obtain the optimal gate location and helps to predict whether there will be weld lines or air traps, sinkholes, and incomplete filling of the part. In this case, a simple 2 mm thick piece with different holes will be used (Fig. 2). The first step is to import the geometry in STEP or IGES format.

Commercial software used for the analysis uses three different ways of meshing a product design. These are Midplane Mesh, Dual Domain Mesh, and 3D Technology. In this case, Tetrahedral elements have been used because the part employed is 3D (Fig. 3).

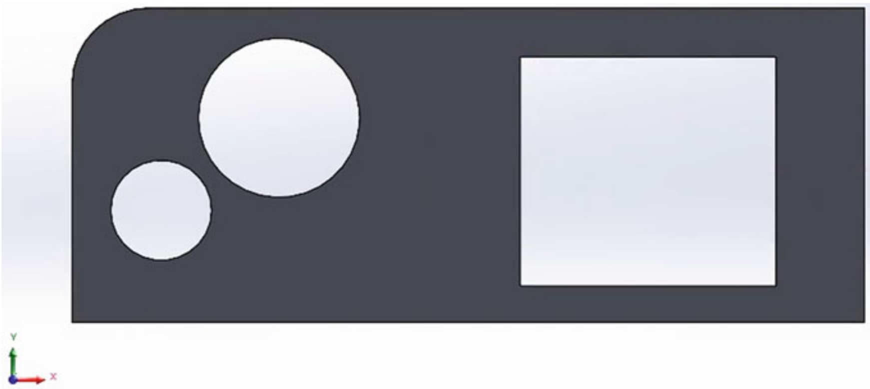

Fig. 2 Part imported from a STEP file

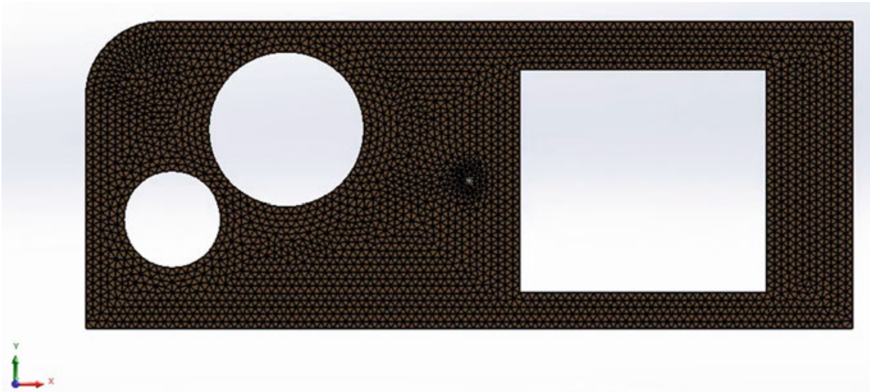

Fig. 3 3D mesh of the part

Optimization of the gate location

To obtain the optimal injection point within the mold cavity, the following will be considered:

- The injection mold geometry.
- An attempt will be made to ensure that the injection point is centered within the mold, to prevent the mold from partially opening during the injection process.
- The position must ensure the complete filling of the part. It will be in areas where the fluid is directed from greater to lesser thicknesses.
- The location must ensure that filling occurs at the same time at all boundaries of the part.

SolidWorks has a tool that helps predict the best entry point location, based on the complete filling of the part and the filling time.

Material and gate dimension

The selection of the material is generally a decision that covers a wide range of possibilities. Today polypropylene is the easiest thermoplastic for the injection process due to its low viscosity, price, low plastic shrinkage, durability, resistance to corrosion, and low water absorption. In this chapter, Generic PP is used, with an injection temperature of 245 °C and a mold temperature of 90 °C.

Not only the entry point of the material into the mold cavity is important, but also the diameter of the gate. Large diameters will fill in less time and with less pressure, but they will leave a visible mark on the piece that can harm its aesthetics. In this sense, a gate of 2 mm is chosen (Fig. 3).

Fill and packing analysis

The main benefit of doing a filling analysis is to predict the fill pattern (Fig. 4). The fill pattern helps identify short shots, which produce incomplete filling of parts. This proves particularly advantageous when dealing with the design of plastic parts featuring varying part thicknesses. Filling pattern analysis also helps to evaluate if the filling process is being carried out in a physical sense. It also partially predicts weld lines and air traps.

Weld lines

Weld line analysis (Fig. 5) is a simple analysis that allows the designer to establish a criterion of whether two plastic faces will form a weak or strong line. Two criteria are generally established:

- The angle at which the encounter occurs. If the two fronts form an angle greater than 135°, it is understood as a welding line; if it is lower, a union line is produced, which has greater mechanical resistance.
- The temperature and pressure at which two plastic fronts encounter occurs. High pressures and temperatures will increase the cohesion of the two plastic fronts, increasing the resistance of the said line.

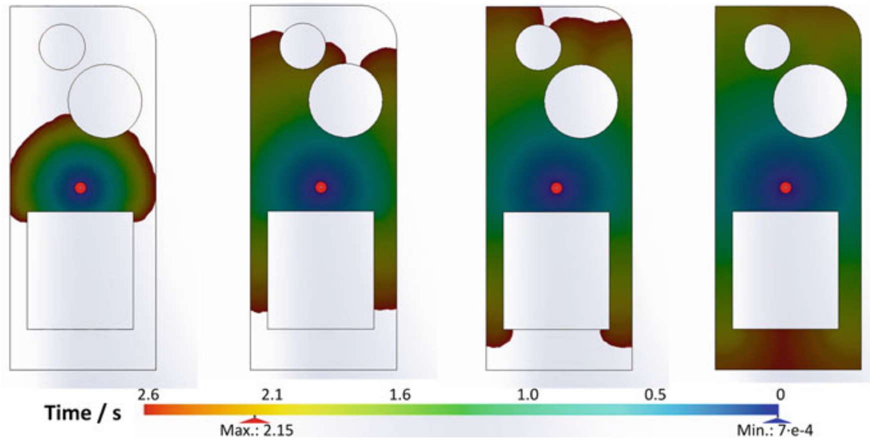

Fig. 4 Fill pattern analysis. The complete filling of the part is done at 2.64 s

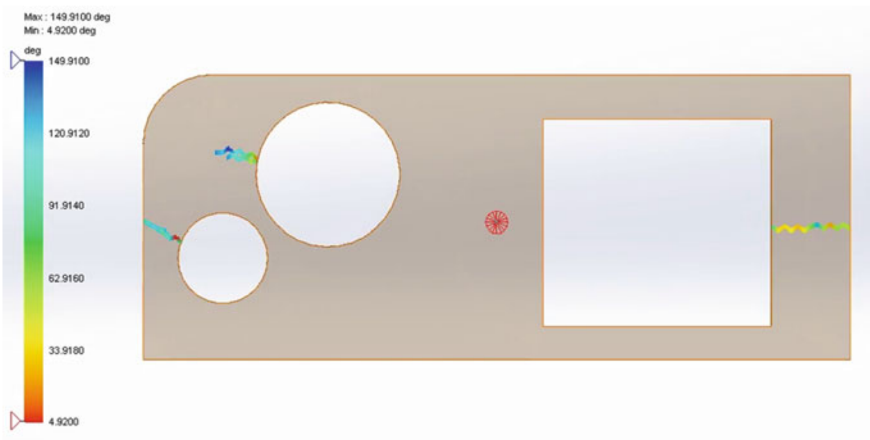

Fig. 5 Weld lines and angles of the plastic fluid fronts

Shrinkage analysis

Although polypropylene has very low typical shrinkage values, it is necessary to analyze them since they can cause important geometric deviations. Shrinkage is the percentage increase in local density from the end of the packing phase to when the part has cooled to the ambient reference temperature (the default value is 25 °C/ 77 °F) (see Fig. 6). Generally, values less than 20% can be admissible.

Clamping force

The clamping force (Fig. 7) is the maximum force required to keep the mold closed during filling. If this is too high or too low, defective parts can result from burners

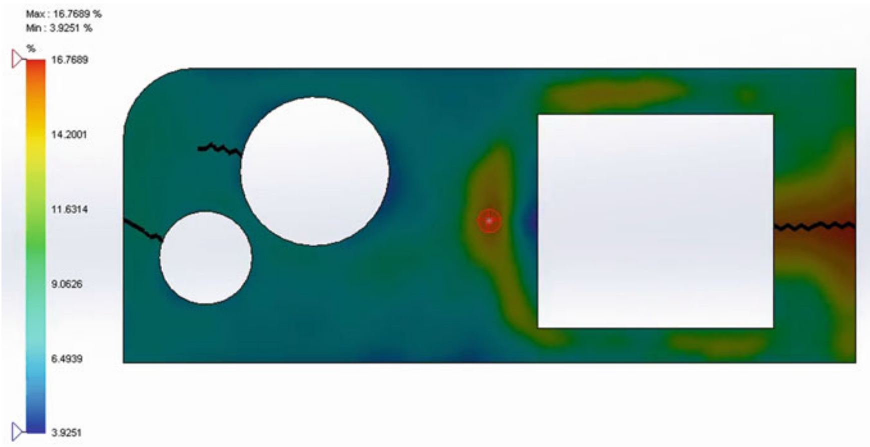

Fig. 6 Shrinkage image after packing

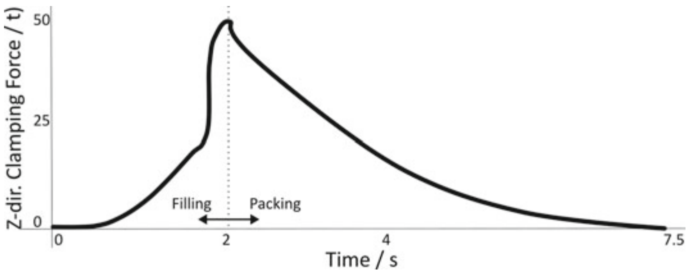

Fig. 7 Clamping force during filling and packing

or burr formation. This force is mandatory to perform an economic analysis of the production. Generally clamping force must be multiplied for 1.2–1.3 times to evaluate the dimension of the injection machine.

6 Outcomes

Expected outcomes of the learning process:

- To identify the important parameters that affect the mold-filling process.
- To understand how the finite element method and its steps can be used to simulate a plastic injection process.
- To be able to design and evaluate the design with a clear orientation towards manufacturing of plastic injected parts.

7 Deliverables and Assessment

Considering the activities described in Sect. 4, each student must submit a report with his/her analysis of the filling exercise. Regarding the assessment, it is proposed a summative evaluation based on the following tools:

- Annotations: participation of the students during the face-to-face activity based on the instructor's annotations. 20% of the score.
- Check list. A checklist is used to evaluate students' reports. 80% of the score.

The report submitted will be evaluated. The main aspects to evaluate are:

- Is the gate location well positioned? Is the part located in the XY plane? Are filling time and full filling of the cavity?
- Evaluation of the feasibility of the injection molding. Is the fill pattern correct? Is the filling time appropriate? Can weld lines be eliminated? and a complete evaluation of the shrinkage.
- Are the clamping force and the packaging well represented?

8 Conclusions

In the analysis of filling feasibility with tools based on the FEM, it is very important to correctly understand the manufacturing process. The simplicity of the analysis can lead the student to think that evaluating the suitability of a part manufactured by injection of thermoplastics is relatively simple. Weld lines, shrinkage, clamping force, and gate location are the most useful outputs for validating the filling process of a plastic part. In this lecture note, a feasibility analysis of mold filling has been proposed, subsequently, outcomes deliverables and assessment are proposed. Students will learn to identify the most important parameters that affect the mold-filling process. In this sense, they will improve their ability in designing plastic parts in injection.