INTRODUCTION

Nowadays, plastic injection molding plays a critical role in the industrial environment and involves a wide variety of applications like packaging, aerospace, construction and automotive [1-4]. Mold design, material characteristics and process conditions are the quality evaluation criteria of the injected parts [5]. Injection mold shape and the process parameters specify the final quality of the injected part. Mold shape is considering an important factor in producing the accurate injected part. Hence, the mold’s characteristics such as gate selection, its position and size must be taken seriously for the conception of the plastic injection mold [7-8]. These characteristics must be carefully designed to ensure a final part without defects and to define significant mechanical properties [8-10].

Researchers have been previously investigated the impact of the mold’s design on the final quality of molded parts. For example, Hassan et al [11] investigated the effect of the gate location on the cooling of injected molded part. They carried out a full three dimensional time-dependent analysis for a mold with cuboids-shape cavity containing two different thicknesses. Furthermore, Vijaykumar et al [12] used a component named to as flow reducer that used to understand design review and mold flow analysis. They applied the software Autodesk, Mold Flow for the conception and the simulation of both the product and the mold. In parallel, Mehdi Moayyedian et al [13] suggested an innovative design of new cross sectional shape of runner system in the plastic injection molding. They looked to find an adequate conception which enable to reduce the cycle time and to scrap easier ejection of runner system from mold tools. Besides, Titomanlio et al [14] examined the shrinkage and the residual stress distributions of the final products and their effect on the injection mold. In addition, Angstadt and Coulter [15] investigated the link between cavity pressure and part quality product in the injection molding process. They considered that the use of cavity pressure as significant potential indicator of the final part quality. In other recent work, Hsu [16] applied the MoldFlow Software to conduct a simulation analysis on aspheric lenses. Also, they adopted the Taguchi Method to optimize the injection molding parameters on the shrinkage of aspheric lenses. Results demonstrated that gate size, holding pressure and filling duration have a considerable impact on the volumetric shrinkage rate. Similarly, Rajalingam et al [17] used Taguchi approach to explore the process parameters generating the shrinkage defect on a plastic cell phone shell component. Optimal molding process parameters are identified using the Design of Experimental (DOE). They consider mold temperature, injection pressure and screw rotation speed as the process parameters in their research study. Also, Jianga et al [18] conducted a study for simulation of injection molding to control volume finite element method. They used local flow information to control time steps for both flow and thermal simulation. Finally, Koya et al [19] examined gate location, diameter and defects through the filling and packing phase of knob that used to set/tune the time for a wash in Washing machine via MoldFlow software and hyper mesh.
From the literature presented above, we can conclude the importance of the numerical simulations in science and engineering for the analysis of mold flowing; and the Taguchi method in solving optimization conditions.

The aim of this work is to study the impact of the mold’s gate size modification on the final quality of the injected PC/ABS parts by integrating two approaches: the Design of Experiment (DOE) using Taguchi and the Computed Aided Engineering (CAE) via SolidWorks Plastics analysis. The DOE has been implemented to select optimal process parameters that could result in a better quality product and shear stresses. Solid Works applied this optimal combination in order to predict the type of defects that could happen following the mold’s gate change.

**EXPERIMENTAL STUDY**

**Material and Mold Design**

The material elaborated is the commercial grade of PC/ABS blend labeled T45 PG. It is widely used in many industrial applications such as packaging, aerospace, construction, and particularly in the automotive field.

According to the BAYER® company, supplier of the PC/ABS blend, the designation of the T45 PG means that it is theoretically composed of 55% ABS and 45% PC. The properties of PC/ABS used in this study are summarized in Table 1 [20].

<table>
<thead>
<tr>
<th>Properties</th>
<th>Method</th>
<th>T45 PG</th>
</tr>
</thead>
<tbody>
<tr>
<td>MVR (260°C/5kg)</td>
<td>ISO 1133</td>
<td>12 cm³/10min</td>
</tr>
<tr>
<td>Melt viscosity(1000s⁻¹/260°C)</td>
<td>ISO 11443</td>
<td>200 Pa.s</td>
</tr>
<tr>
<td>Vicat softening temperature</td>
<td>ISO 306</td>
<td>112°C</td>
</tr>
<tr>
<td>Tensile strength (50mm/min)</td>
<td>ISO 527</td>
<td>49 MPa</td>
</tr>
<tr>
<td>Izod notched impact strength (23°C)°</td>
<td>ISO 180</td>
<td>40 kJ/m²</td>
</tr>
</tbody>
</table>

The injected molded PC/ABS parts were produced in the SKG factory. It is a local company specialized in the injection and electroplating of the PC/ABS automotive parts, established in Sfax, Tunisia.

To produce suitable specimens (Figure. 1), a mold was designed (Figure. 2) and fabricated in the SKG company. As illustrated in Figure. 2(a), three cavity imprints were elaborated. During the plastic injection molding process, the tolerance of the molded part has been considered.

The originality of using this plastic injection mold is in producing three mechanical test specimens: Tensile (Eₜₐ), Dynamic tensile (Eₜ₉) and torsion (E₉₉) samples as illustrated in Figure 1 (a). The tensile test specimens conform to the standard ISO 527-2. The dynamic tensile samples (Eₜ₉) conforms to the standard ISO 8256. Mechanical specimens dimensions are given in Figure 1(b). Each specimen has a specific geometry different from the others. This difference will be marked in the molten polymer flows at the end of the filling phase.

![Figure 1. The obtained specimens (a) and their dimensions(b) in mm.](image-url)
Figure 2(a) illustrates the plastic injection mold considered in this investigation. The actual gate size of the injection mold is depicted in Figure 2(b). The gate or also called the orifice of molten material into the cavities of the injection mold is one of the most important design variables in mold design. It will affect filling time, temperature and pressure distributions. The edge gate is the type considered in this investigation. This type is in agreement with J.P. Beaumont [21] who reported that the edge gate is considered the most basic that has the rectangular cross section whose main advantage is for use in multi cavity molds.

**Figure 2.** The injection mold (a) and the actual gate size in mm (b).

**Numerical Simulation Protocol**

Optimal process parameters found via Taguchi approach using the actual mold’s gate size are depicted in Table 2 [20]. This optimal combination yielded best shear stress with no injection defects.

<table>
<thead>
<tr>
<th>Process parameters</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material temperature</td>
<td>260 °C</td>
</tr>
<tr>
<td>Injection pressure</td>
<td>50 bar</td>
</tr>
<tr>
<td>Holding Time</td>
<td>8 sec</td>
</tr>
<tr>
<td>Mold temperature</td>
<td>60 °C</td>
</tr>
</tbody>
</table>

**Table 2.** Optimal process parameters.

Figure 3 presents molded PC/ABS part elaborated according to the optimal process combination. It confirmed effectively that there is no defect observed on the surface of the injected part.

**Figure 3.** Flawless surface for the optimal injection parameters.
Such modification in the mold could affect the way in which the molten polymer flows into the mold cavity. Therefore, a reduction up to 50% in the mold’s gate was proposed by the SKG Company to explore the behavior of the material and to find out the defect occurred by applying the same optimal parameters used with the actual gate size.

Our work is in agreement with some scholars like W. Dai et al [22] and D.A.R. Calhoun et al [23] who revealed that the gate dimension should be as small as possible to avoid the occurrence of visible blemishes on the part’s final quality and excess material removal. Similarly, Li et al [24] applied a methodology for single gate location optimization for plastic injection mold in order to optimize the warpage via numerical simulation software MPI. In parallel, Shen et al [25] established a new methodology that predicts the optimum gate location of injection molds based on injection molding simulation. Also, M. Moayyedian et al [7,8] who changed the size and geometry of runner and gate system in order to reduce the scrap rate of injected parts.

**SIMULATIONS**

**Gate Modification Analysis**

The gate size of the mold is reduced to 50% in terms of dimensions from that of the actual gate size. This modification resulted from a slick entering of the molten polymer blend into the cavities by creating flow direction during the injection process.

The mold’s characteristics are defined as follow: the injection point Ip and the dimensions of the actual and the proposed gate size change in the Figure. 4 (a) and Figure. 4 (b), respectively.

![Figure 4](image-url)  
**Figure 4.** The actual (a) and the proposed (b) gate size modification in mm.

**Simulations Results**

The optimal injection molding parameters are defined according to the material blend and the injection machine used. Material temperature of 260 °C, injection pressure of 50 bar, mold temperature of 60 °C and holding time of 8 sec are the adequate parameters for this simulation. The type of meshing is cubic. The cycle time, cooling time, internal defects such as residual stresses will be affected by the new gate system dimension.

Ease of filling, filling time analysis, weld and meld lines analysis and injection pressure at the end of filling are the key factors to evaluate the suggested edge gate size.

As depicted in Figure. 5, the ease of filling level is ranging between 0 and 2 with three colors. Red color refers to a bad level of filling, yellow color for the average level and the green color for the best level of filling. Both of the existing and the proposed gate size system reached the most satisfactory level represented by the green color.
Another chief factor to verify the efficiency of the new proposed gate system is the filling time. The difference in term of filling time between the existing and the new gate size system is negligible. This finding is proved via the numerical simulation depicted in Figure 6. We can conclude that both injected part are well filled within an acceptable filling time.

Researchers like Li, Chen et al [26] stated that the short shot defect would happen during the injection molding exactly when there are long flow distances or on thin walls and far from the gate. This situation was sensibly avoided especially that the holding time in this simulation was 8 sec.

The weld lines defect is observed in the new suggested gate size near to the dynamic tensile and the torsion specimens as illustrated in Figure 7(b), which is not the case for the actual gate system (Figure. 7(a)).

The proposed modification on the gate size system induced the occurrence of this blemish. Weld lines represent a threat in the plastic industry since they cause to weaken or/break of the injected molded parts. This defect is the result of the thin and solidified layers of the fronts of the molten polymer flow paths meet and fuse, before solidifying again with the rest of the plastic. Therefore, the occurrence of such defect could cause a serious structural and appearance problems of the injected parts [27, 28].
The residual stresses are the stresses lasting inside the molded part under the condition of no external loads. They are solidified inside the part in the molding process. These internal defects are the principal reason of part shrinkage and warpage.

Some scholars investigated the effect of these blemishes. For instance, Östergen [29] defined two types of residual stress; namely, flow-induced residual stress and thermal-induced residual stress. This author and Erzurumlu and Ozcelik [30] agreed that the existence of these defects in injected parts present a serious problem meanwhile their high values may lead to self-released cracking of the mold pieces and their deformation.

As shown in Figure. 8, the residual stresses were spread in all the injected PC/ABS parts. The runner system and the bottom part of the tensile specimen closest to the injection point Ip are the mostly attacked by these internal defects. Besides, residual stresses basically have a great impact on the strength properties of the injected PC/ABS parts. They may remain fixed inside the molded specimens and may cause a major transformation in their dimensions.

The inlet maximum pressure is considered one of the most significant criterions for the evaluation of the part geometry. It is clearly seen from Figure. 9 that injected parts of the existing and the new proposed gate system are filled successfully with an injection pressure of 2.55 and 4.75MPa, respectively.

This pressure is considered high compared to the actual mold’s gate size. This finding confirms that the modification of the mold cavity generates certain defects and slows down the filling process and subsequently increases the pressure at the inlet.
CONCLUSION

Computer Aided Engineering (CAE) via SolidWorks Plastics simulation is successfully used to determine the defects generating following the mold’s modification. Optimal injection process parameters including a material temperature of 260 °C, an injection pressure of 50 bar, a holding time of 8 s and a mold temperature of 60 °C confirmed by Taguchi are tested numerically.

Results prove that the modified gate system in comparison with the actual gate has more turbulences of molten polymer flows into the cavities, which leads to the occurrence of some internal and external defects in the final injected parts such as weld lines, residual stresses and high inlet pressure. These blemishes will demean the final quality of the injected PC/ABS part and decline the metallization process. The experimental set up will be conducted as future work to validate the simulation result.

REFERENCES