Injection Moulding Recommended Parameters Simulation Analysis of Rice Husk Composite

Haliza Jaya¹, Nik Noriman Zulkepli³, Mohd Firdaus Omar²*, Shayfull Zamri Abd Rahman³, and Khairul Anwar Abdul Halim²

¹Faculty of Engineering Technology, Universiti Malaysia Perlis (UniMAP), 01000, P.O. Box, D/A Pejabat Pos Besar, Kangar, Perlis, Malaysia
²School of Materials Engineering, Universiti Malaysia Perlis (UniMAP), Kompleks Pengajian Jejawi 2, 02600 Arau, Perlis, Malaysia
³School of Manufacturing Engineering, Universiti Malaysia Perlis (UniMAP), Kampus Alam Pauh Putra, 02600 Arau, Perlis, Malaysia

Abstract. Study of recommendation parameters for injection moulding process was carried out with aid of Autodesk Moldflow Insight. Rice husk filled low-density polyethylene was used as the raw material. Standard dog-bone shape mould with ASTM were used throughout the simulation analysis. The investigations were carried out on fill analysis, fill + pack analysis and cool analysis on injection moulded dog-bone shape. From the simulation analysis, the recommended parameters of the injection moulding process for rice husk composites were determined.

1 Introduction

At this date, various field of industries interested in injection moulding has either read the power of flow analysis for injection moulding, or used the technology to aid in the design of injection molded products [1]. It is commonplace for new designs to be numerically simulated or even optimized via computer simulation before the first piece of steel is cut for the mold. Rather than expand time and money building a series of prototypical tools to fine-tune a new product, computer simulation has dramatically affected the way new parts are designed. Designers and process engineers have effectively leveraged the power of simulation to shorten the design process and to eliminate processing problems that once caused reductions in quality and profitability.

Computer simulations offer the tremendous advantage of enabling designers and engineers to consider virtually any molding option without incurring the expense associated with material waste and machine time. The ability to try out new designs or concepts on the computer gives the engineer the opportunity to eliminate problems before beginning production [2]. In addition, the engineer can quickly and easily determine the sensitivity of specific molding parameters on the quality and production of the final product. This computer aided engineering (CAE) virtual design offers tremendous flexibility to the engineer to determine the effects of different gating scenarios, different runner designs, geometric

* Corresponding author: firdausomar@unimap.edu.my
features, and different molding and processing conditions on the moldability and quality of the final product [3]. Furthermore, these CAE tools allow these virtual designs to be completed expediently in a matter of days or even hours rather than the weeks associated with experimental trial and error. Commercially available simulation had been available as early as the mid-1970s in the form of time-shared computer simulation using a lay-flat approach. By the late 1980s a number of different companies such as Moldflow, C-MOLD, and Simcon offered simulation tools for CAE in injection moulding [4]. In addition to the commercially available packages, a number of research codes were developed by universities and independent companies.

2 Methodology

Autodesk Moldflow Insight (AMI) software was used for the simulation process. The purpose of simulation process is to determine the recommended value for the related parameters in injection moulding process. Some of the parameters were selected as an input variable and the others were fixed at the recommended setting. The ranges of input variables were also determined at this stage. The simulation process started by importing a part, a gating system and a cooling line design from modelling software and analyzed based on the process setting that has been set up in the software. In simulation analysis, three analyses were conducted such as fill analysis, fill + pack analysis and cool analysis. All recommended injection moulding parameters were obtained from related result of simulation analysis. The input parameters used in each analysis are listed in Table 1 to obtain the recommended parameters of the thick plat part.

Table 1. List of parameters used in simulation analysis

| Analysis        | Input parameter                  | Output parameter       |
|-----------------|----------------------------------|------------------------|
| Fill analysis   | Material properties              | Injection profile      |
|                 | Mould temperature (°C)           |                        |
|                 | Melt temperature (°C)            |                        |
| Fill + Pack analysis | Material properties           | Packing pressure (MPa) |
|                 | Injection profile                |                        |
|                 | Mould temperature (°C)           |                        |
|                 | Melt temperature (°C)            |                        |
| Cool analysis   | Material properties              | Cooling time (s)       |
|                 | Mould temperature (°C)           |                        |
|                 | Melt temperature (°C)            |                        |
|                 | Coolant inlet temperature (°C)   | Packing time (s)       |

2.1 Import 3D part and gating system

The first step in the simulation process involved importing the part and gating system into AMI software. Two cavities mould were selected according to ISO 3167:2002 (E) for thick flat parts as shown in Fig. 1. Next, the part and gating system were meshed.
Fig. 1. Thick flat parts with gating system

2.2 Meshing part and gating system

The part and the gating system were meshed in 3D Mesh to obtain more accurate results as shown in Fig. 2. As many as 345914 tetrahedral elements were built in part and gating system, that are connected with 64123 nodes.

Fig. 2. Meshed model for part and feed system

2.3 Moulding window analysis

Moulding window analysis was used to determine the recommended mould and melt temperatures. A single cavity was required to evaluate the quality of the moulded part based on the material selected as shown in Fig. 3.

Fig. 3. Single part used in moulding window analysis
2.4 Create cooling circuit for core and cavity

Cooling circuit for core and cavity inserts were designed and modelled as shown in Fig. 4. Cooling lines were created using beam elements with 6 mm diameter. Eighty eight beam elements were created for cooling line that is connected using 90 nodes.

![Fig. 4. Cooling circuit design](image)

2.5 Fill and Pack analysis

The input parameters for injection moulding process were set in process setting taskbar. These settings were used as a guideline to calculate the results. The parameters such as melt temperature, packing pressure, mould temperature, velocity/pressure (V/P) switch-over, packing time, and injection profile were set in the process setting. Besides, material properties of mould and injection moulding machine specification were also set in this window.

3 Results and Discussion

3.1 Fill analysis results

The fill analysis was conducted to observe the flow of molten plastic into the mould cavities from the gating system. A fill analysis based on the recommended setting of mould and melt temperature, automatic setting in filling control, V/P switch-over and default packing/holding control, was conducted without consideration on the cooling channel designed. The filling time (s) was observed in this analysis and the results are shown in Fig. 5. The fill time and maximum shear rate obtained in this analysis were used as a reference to determine the filling velocity profile. In addition, the maximum shear rate should not exceed the maximum shear rate allowed in material database. Shear rates are important to the design and moulding of plastic products. As shear rates increase beyond the allowable maximum shear rate, it will increase the stress towards the molecular chains that make up the polymer and eventually damage the molecular chains apart. This will lead to the poor mechanical properties of a product [5].
3.2 Fill + Pack analysis results

The packing pressure evaluation was done based on the fill analysis results. From fill analysis results, the maximum injection pressure to fill the cavities is 18.6 MPa. The mass of parts depicted in Fig. 6 were evaluated by increasing the packing pressure until none or only small changing of the part mass occurred. Part mass shows slight increase with the increase of packing pressure until 16.11 MPa. Part mass remained 14.1 g even after 18.9 MPa packing pressure was used. This result shows that 16.11 MPa packing pressure was adequate for the moulded process.

| Pack Analysis  |
|----------------|
| Time (s) | Packing (%) | Inj Press (MPa) | Clamp Force (tonne) | Part Mass (g) | Frozen (%) | Status |
| 5.619 | 0.80 | 1.511e+01 | 5.41e+00 | 1.39e+01 | 18.61 | P |
| 5.631 | 0.94 | 1.511e+01 | 5.91e+00 | 1.39e+01 | 18.60 | P |
| 5.653 | 0.88 | 1.511e+01 | 6.51e+00 | 1.39e+01 | 18.65 | P |
| 5.693 | 0.16 | 1.511e+01 | 7.29e+00 | 1.39e+01 | 18.76 | P |
| 5.746 | 0.27 | 1.511e+01 | 7.89e+00 | 1.39e+01 | 18.90 | P |
| 5.825 | 0.41 | 1.511e+01 | 8.40e+00 | 1.39e+01 | 11.22 | P |
| 5.992 | 0.74 | 1.511e+01 | 8.65e+00 | 1.39e+01 | 11.95 | P |
| 6.568 | 1.15 | 1.511e+01 | 8.75e+00 | 1.48e+01 | 14.42 | P |
| 7.211 | 1.38 | 1.511e+01 | 8.45e+00 | 1.48e+01 | 19.18 | P |
| 8.923 | 6.30 | 1.511e+01 | 6.24e+00 | 1.48e+01 | 23.87 | P |
| 9.552 | 7.60 | 1.511e+01 | 2.80e+00 | 1.48e+01 | 26.57 | P |
| 10.716 | 9.85 | 1.511e+01 | 2.97e+00 | 1.48e+01 | 31.28 | P |
| 11.941 | 12.23 | 1.511e+01 | 1.80e+00 | 1.48e+01 | 36.22 | P |
| 13.179 | 16.60 | 1.511e+01 | 1.80e-02 | 1.48e+01 | 39.90 | P |
| 14.365 | 17.85 | 1.511e+01 | 9.11e-02 | 1.48e+01 | 45.41 | P |
| 15.576 | 19.22 | 1.511e+01 | 8.92e-02 | 1.48e+01 | 47.79 | P |
| 15.577 | 19.23 | 1.400e+01 | 7.95e-03 | 1.48e+01 | 47.79 | P |
| 15.582 | 19.25 | 6.840e+00 | 8.10e-03 | 1.48e+01 | 47.81 | P |
| 15.586 | 19.25 | 8.000e+00 | 1.22e-03 | 1.48e+01 | 47.87 | P |
| 15.601 | 19.27 | 8.000e+00 | 1.21e-03 | 1.48e+01 | 47.87 | P |

Fig. 5. Result fill time from fill analysis: 5.498 s

Fig. 6. Part mass in Pack analysis
3.3 Cool analysis results

Packing time and cooling time were determined in this section. Cool Finite Element Method (FEM) analysis was run with the consideration of cooling channel design and coolant inlet temperature according to the software recommendation. The result of cooling time of the moulded part was obtained from time to time to reach the ejection temperature as shown in Fig. 7. The recommended cooling time is 12.03 s while the recommended time for packing is 10.3 s. From the Cool analysis results, packing time can also be identified. Gate is the smallest area in the mould that the first region will be frozen. When the gate freezes, there are no material allowed to enter the part. Therefore, the packing time is equal to the time of gate freeze as portrayed in Fig. 8.

![Figure 7. Recommended cooling time : 22.32 s](image-url)
Fig. 8. Recommended packing time: 19.02 s

Based on the simulation analyses, the recommended parameters for the injection moulding process were summarized in Table 2 according to AMI 2012 software. These results were used as a reference in the next step of this study. All the values in Table 2 are satisfy the requirement of injection moulding machine used in this study based on the LDPE properties.

| Parameter                        | Recommended value |
|----------------------------------|-------------------|
| Mould temperature (°C)           | 50                |
| Melt temperature (°C)            | 170               |
| Fill time (s)                    | 5.498             |
| Packing time (s)                 | 19.02             |
| Cooling time (s)                 | 22.32             |
| Initial ram speed (mm/s)         | 8.237             |
| Final ram speed (mm/s)           | 8.318             |
| Initial ram position (mm)        | 66                |
| Second ram position (mm)         | 50.4              |
| P/V switch-over position (mm)    | 17.3              |
| Cushion (mm)                     | 5                 |
| Packing pressure (MPa)           | 16.11             |

**Table 2. Summary of recommended injection moulding parameters via simulation**

**Conclusion**

Three types of analysis were simulated to obtain the recommended setting of machine parameters for moulding the LDPE/rice husk composites parts. Fill analysis was conducted to obtain the recommended filling profile on the molten plastic into the mould cavities. Then, Fill+Pack analysis was performed to determine an appropriate packing pressure of the
moulded part. Finally, the packing time and cooling time were observed in Cool Analysis. The results of the recommended parameters were discussed and compiled in Table 2. Therefore, the objective of this study which is to determine the recommended moulding parameters of the LDPE/rice husk composites parts has been achieved.

References
1. B. Benhadou, A. Haddout, M. Benhadouand, H. Bakhtari. Int. J. Mech. Eng. Technol., E 7 (2016)
2. Z. Zhang and B. Jiang. J. Wuhan Univ. Technol. Sci., E 22 (2007)
3. A. Rios, B. Davisand, P. Gramann. Proc. Compos. Fabr. Assoc. Tampa (2001)
4. Turng, L. S., M. Peić. Proc. Ins. Mech. Eng. B J. Eng. Manuf. E 216 (2002)
5. J. Shoemaker, Hanser Verlag. E 10, 153 (2006)