*Sensors and Materials*, Vol. 34, No. 6 (2022) 2243–2252
MYU Tokyo


S & M 2969

# Design of Injection Molding of Side Mirror Cover

Chi-Hung Lo,[1*] Cheng-Chung Chen,[2]
Hwee Ling SIEK, Perline,[3] and Clara Metha Yoandara[1]

[1]Department of Industrial Design, Tunghai University,
No. 1727, Sec. 4, Taiwan Boulevard, Xitun District, Taichung, Taiwan
[2]Department of Hospitality Management, Tunghai University,
No. 1727, Sec. 4, Taiwan Boulevard, Xitun District, Taichung, Taiwan
[3]De Institute of Creative Arts and Design , UCSI University, UCSI Heights,
1, Jalan Puncak Menara Gading, Taman Connaught, 56000 Cheras, Wilayah Persekutuan Kuala Lumpur, Malaysia

(Received December 30, 2021; accepted April 7, 2022)

***Keywords:*** motor spare parts, lifter mechanism, rearviewside mirror, mold construction design

We designed an injection molding process based on the product analysis of the cover of a motorcycle side mirror. The purpose is to develop a design for the injection molding of the product, which is applied in the concept of Industry 4.0 that aims to have intelligent processes. We considered the clamping force, venting and cooling, gate, runner, and lifter for manufacturing a cover of a motorcycle side mirror without defects, warpage, and undercuts, but with an efficient process with the help of various sensors. The final design of the mold of the cover included lifters with an opening of 250 mm and a height of 630 mm. Multiple pressure and temperature sensors can be installed in the gate, runner, and cavity of the mold for manufacturing defectless products. The cycle time of the molding process was set to 20.5 s. The proposed design meets production needs in an effective time for manufacturing and leads to developing the designs of similar products.

## 1. Introduction

The use of plastics in parts and components of devices and machines is common even for vehicles. As functionality and appearance need to be considered, the parts of vehicles, such as mirrors, body covers, and lamp covers, require various forms and strengths depending on their functions. The common material for manufacturing such parts is plastic owing to its ease of use in shaping, elasticity, lightweight, and low cost[(1)]. Injection molding is a general method of producing plastic products for various purposes.

The injection molding of plastics is divided into four steps: filling, pressurizing, cooling, and demolding. In the filling process, raw material is melted and injected through a runner into a mold cavity. When the cavity is filled with melted plastic, high pressure is applied to remove the shrinkage effect of the plastic. Then, the molded plastic is cooled to be demolded.[(2)] Demolding is important as the optimal design of final products is seen in the demolded product. Techniques

*Corresponding author: e-mail: chlo@thu.edu.tw
https://doi.org/10.18494/SAM3828

ISSN 0914-4935 

and designs of the molding process are closely related to the demolding process and address a crucial part of the final product's design.

For injection molding, the following need to be considered: mold base, injection method, appropriate pressure, cooling system, and the design of the final product.(3) The quality of molded products depends on several factors such as the flow rate of the injection of melted plastic,(4) the geometric calculation of injection molded parts,(5) and the mechanical properties of the material used such as shrinkage value,(6,7) the viscosity of the melted material,(8) and the removal of gas during injection molding.(9) Although these are considered important factors for injection molding, the quality is determined mainly by the appropriate design of the runner and lifter of the injection molding system. The runner is used to quickly fill the cavity of the mold to minimize defects such as sink marks and warpage,(10) while the lifter is used to form the undercuts of a molded plastic part and eject it out of the mold.

Injection molding in the era of Industry 4.0 requires precise process control that considers diverse factors and parameters. The process control demands many sensors in the molding process, such as pressure, temperature, and optical sensors. Optical and ultrasonic sensors may also be needed in the process.(11) However, the appropriate use of the sensors needs to be accompanied by the proper design of the injection molding process. Therefore, injection molding requires a sophisticated design by considering the above-mentioned parameters and the proper installation of the necessary sensors. Thus, it is necessary to have an efficient way of designing the injection mold by considering the diverse elements and factors of the process, which leads to the manufacturing of high-quality final products. Therefore, we propose an injection molding design for the cover of a motorcycle mirror as an example of a design process that fits into Industry 4.0.

## 2. Materials and Methods

### 2.1 Design and specifications

For complex undercuts or 3D hollow molding, two different manufacturing techniques are used: shell and fusible-core techniques. We adopt the shell technique because the cover of a motorcycle mirror has snap-fits as a lock. We apply a cantilever snap-fit to the cover as it is the most frequent type of snap fit. The cantilever can be a straight bar or an L- or U-shaped lever. If it is placed at the edge of gaps, it does not require sliders in injection molding.(12)

We assume acrylonitrile butadiene styrene (ABS) as the material for the cover for mass production. The final product needs to have parting line marks of a lifter, a core cavity on the top of the outer side, and a snap-fit in the product. The product has a height of 25.12 mm, an outer diameter of 108.2 mm, and a dominant thickness of 2 mm. Snap-fits are included as a lock for the holder of a mirror. The design of the cover is shown in Fig. 1.

This product requires a smooth surface with a smoothness equivalent to N4 on the outside and N5 on the inside. The product has four snap-fits as connections in the mirror holder. (Fig. 2)

The specifications of the cover are important for designing the cavity and core of the mold as they affect the life of the cover. The product specifications are shown in Table 1.

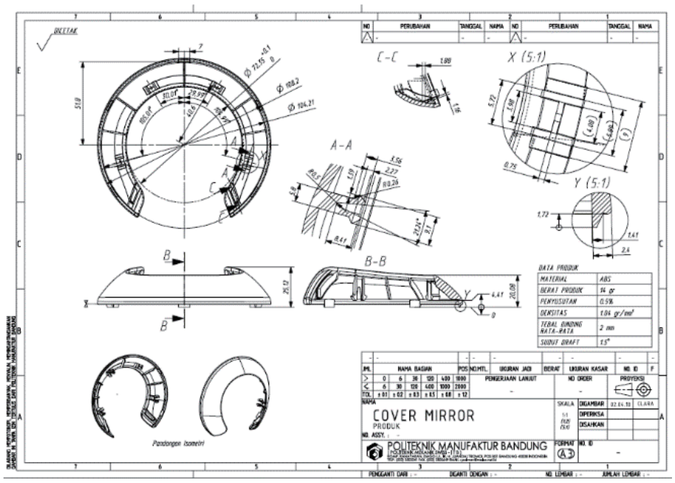
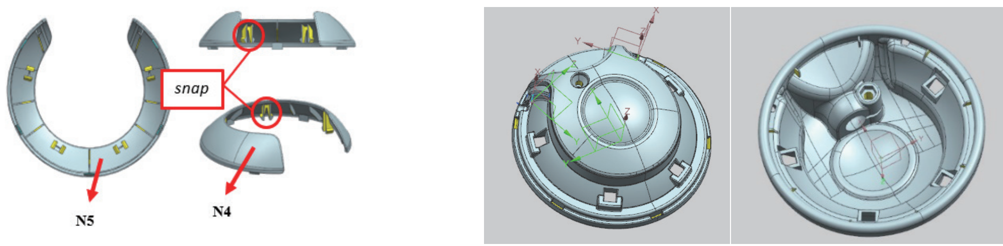



Fig. 1. Design of cover of a motorcycle side mirror.




(a) (b)

Fig. 2. (Color online) Design of injection mold of cover of a motorcycle side mirror.

Table 1
Product specifications.

| | |
|---|---|
| Function | Cover styling on rearview mirror |
| Diameter | $108.2 \times 25.12$ mm$^2$ |
| Material | Acrylonitrile butadiene styrene (ABS) |
| Shrinkage | 0.50% |
| Angle | 1.5° |
| Density | 1.03 g/cm$^3$ |
| Volume | 13502.70 mm$^3$ |
| Weight | 13.9 g |
| Process | Injection mold |
| Outer appearance | Painted |

## 2.2 Calculations for design

To obtain the optimum number of cavities, the theoretical number of cavities is calculated on the basis of the entire size, diameter, and estimated displacement of the cover, and the plasticizing rate of the mold system. After the number of cavities is calculated, the layout of the cavities relative to the central sprue is finalized to meet the following conditions:

(1) The filling time of all the cavities needs to be decided so that the injected material maintains the same temperature.
(2) The flow length needs to be minimized for minimum scraps.
(3) The distance of each cavity has to ensure enough space for cooling and ejector pins. An adequate cross-sectional area is required to withstand the injection pressure.
(4) All the forces are required to be in the center of gravity.

The minimum clamping force is determined from the reactive force of the cavities by considering the projected area of all the cavities and the maximum cavity pressure in the runner.

### 2.3 Injection control

According to the estimated clamping force, the highest clamping force is found for the optimal control of injection molding. Then, the parting lines of the product are decided as the opening direction of the mold. The parting lines are determined by undercut features, mostly by the edge shape of the product.[5] Plastic mold products have visible parting lines. Thus, the positions of the lines are important in terms of appearance and functions. The cover has the lines at the edge of the injection molding opening outside of the product, which separates the insert core and cavity as shown in Fig. 3.

## 3. Results and Discussion

### 3.1 Design of snap-fits

From the product requirements, the most crucial features are the snap-fit joints, which are the main subject of this study. The type of snap-fit on this product is a cantilever, which is the most common snap-fit in product design (Fig. 4). The strength of the snap-fit joint requires adjustment according to the ejection mechanism and is calculated as[6]

$$y_{permissible} = 1.09 \times \frac{\varepsilon \cdot l^2}{h}, \tag{1}$$

where $y$ is a permissible deflection, $\varepsilon$ is the permissible strain, $l$ is the length of the arm, and $h$ is the thickness. The allowed deflection is 1.65 mm with $y$ of 0.28 mm. $y$ is smaller than the allowed deflection as it requires a lifter as an ejection helper when ejection is not forcefully carried out.

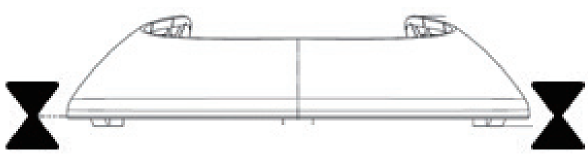


Fig. 3. Parting lines of cover of motorcycle side mirror.

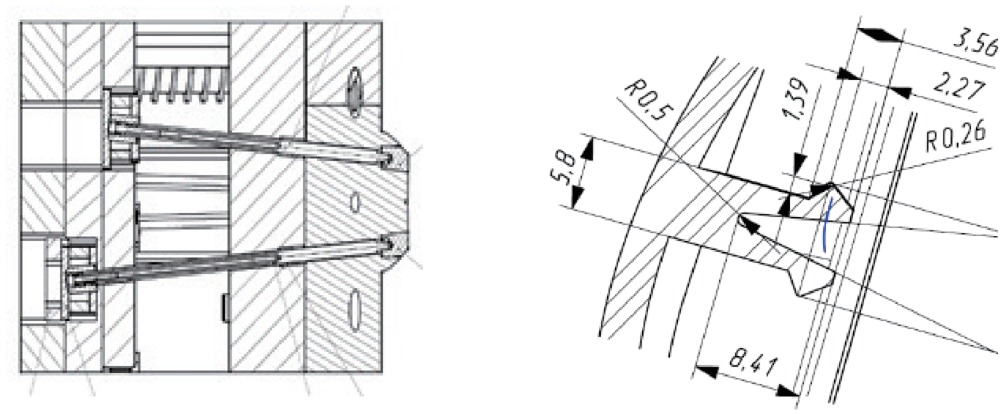



Fig. 4. Design of lifter and snap-fit of cover of motorcycle side mirror.

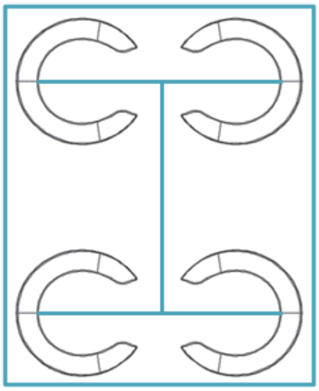


Fig. 5. (Color online) Cavity layout of cover of motorcycle side mirror.

## 3.2 Design of cavity layout

Figure 5 shows the layout of the cavities of the mold. As ABS shrinks by 0.5%, the cavity size needs to be larger than the actual size of physical product because plastic has the characteristic to shrink after the injection time.(7) This mean that the cavity needs to be upsized 0.5% to produce the precise desired dimensions of the actual product. The cover of the motorcycle mirror has an insert cavity, an insert core, and a lifter. Designing the cavity layout requires the consideration of the shape of the final product, including the parting lines and ejection system, and machining.

## 3.3 Clamping force

During the injection molding process, the plastic liquid fills the mold cavity, and pressure is exerted in all directions to push the mold cavity. Clamping force is the force required to withstand internal pressure so that the mold cavity does not open during the injection process. The required force is determined by considering the exerted pressure, the thickness of the mold wall, the projection area of the cavity of the mold, and the position of the gate of the mold. We used the Demag and Tat Ming methods to calculate the clamping force.(3,13)

The Demag method is based on the wall thickness diagram, flow path, and difference in internal pressure. We adopted the following equation for the calculation:

$$Fc = p_{spec} \times A_p \times n \times f_v \,, \tag{2}$$

where $p_{spec}$ is the specific internal pressure, $A_p$ is the projection area of the product, $n$ is the number of cavities, and $f_v$ is the viscosity factor obtained from the ABS viscosity factor table (the highest viscosity factor is 1.4).(8,13)

To find the specific internal pressure, we applied the path-wall thickness ratio to the specific internal pressure diagram shown in Fig. 6. Since we used a 70:1 ratio and the wall thickness was 2 mm, the specific internal pressure was 120 bar.

The specific internal pressure was gained from wall thickness by using the following equation:

$$p_{spec} = Lp \times sf, \tag{3}$$

where $Lp$ is the length of the flow path and $sf$ is the wall thickness safety factor (Table 2).

As the measured internal pressure was less than 100 kg/cm², we assumed that the maximum specific internal pressure was 120 kg/cm². This led to the clamping force of 27.48 tons by the Demag method.

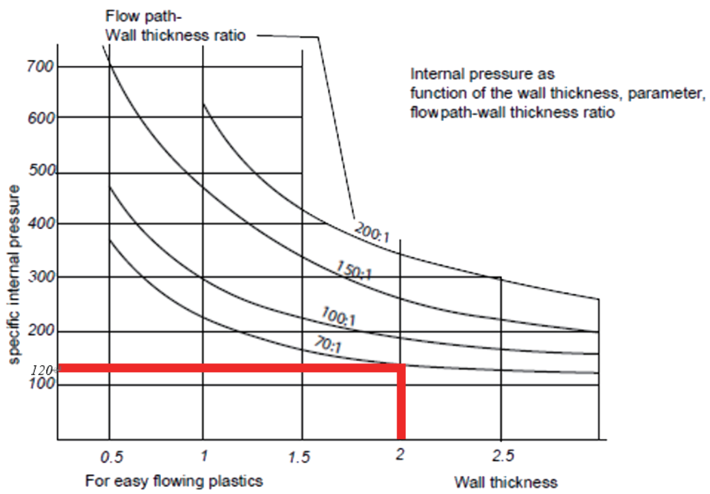



Fig. 6. (Color online) Diagram of flow path-wall thickness ratio according to specific internal pressure.(2)

Table 2
Wall thicknesses and safety factors.(2)

| $T$ (mm) | 0.5 | 0.6 | 0.7 | 0.8 | 0.9 | 1 | 1.1 | 1.2 | 1.3 | 1.4 | 1.5 | 1.6 | 1.7 | 1.8 | 1.9 | 2.0 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| $sf$ (kg/cm$^3$) | 100 | 70 | 57 | 45 | 35 | 30 | 26 | 21 | 18 | 15 | 12 | 11 | 10 | 9 | 8 | 7 |

The other method that we used for the calculation is the Tat Ming method. To calculate the clamping force, the length of the plastic flow from the gate to the farthest end of the cavity of the mold is required.(3,5) The measured flow path (26.01 cm) was used to calculate the clamping force using the following equation:

$$Fc = e \times A_p \times n \times f_v \,, \tag{4}$$

where $e$ is an empirical pressure factor from the empiric table (0.388–0.62 ton/cm² for ABS),(13) $A_p$ is the projection area of the product, $f_v$ is the viscosity factor of ABS, and $n$ is the number of cavities. We chose the smallest empirical factor by assuming a short flow path and a large product thickness. The estimated clamping force was larger than 88.86 tons.

For exerting an appropriate clamping force, multiple pressure sensors are required to monitor the internal pressure. The transmitted data of the internal pressure helps the molding machine decide how much pressure to put on the clamps.

### 3.4 Core and cavity of mold

To determine the dimensions of the core and cavity of the mold, the shrinkage factor of the injected material must be considered. ABS has a shrinkage of 0.5%,(13) which was added to each dimension of the core and cavity.

### 3.5 Gate and runner

The gate is a hole for injecting melted plastic into the cavity of the mold. The position and size of the gate affect the quality of the product in terms of both appearance and dimensions. Thus, the shape and dimensions need to be considered carefully. We chose edge gates for the mold.(2) The runner is a channel through which the plastic flows from the gate to the cavity. Its diameter depends on the type of material used. Figure 7 shows the shape of the gate and runner.

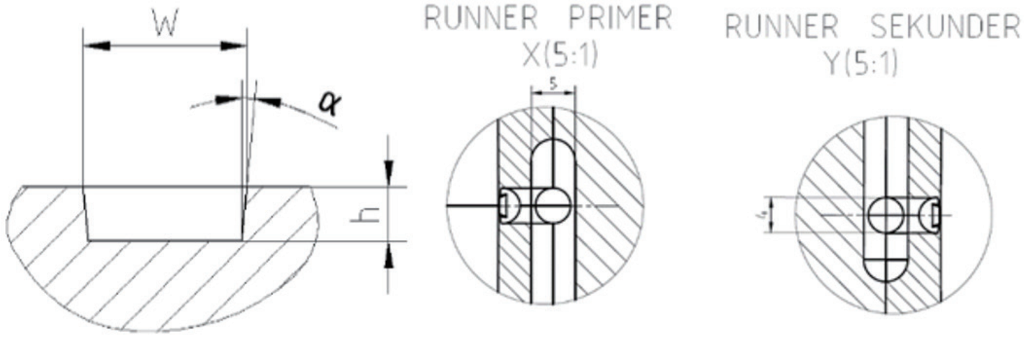


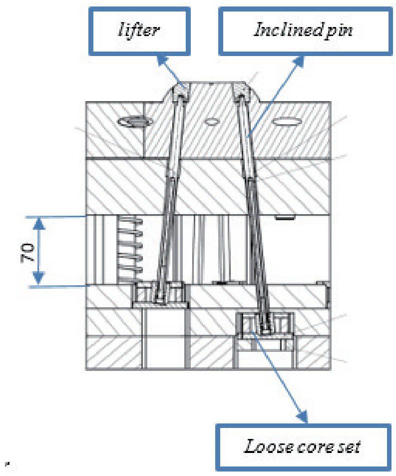

Fig. 7. Design of gate and runner of mold of cover of motorcycle side mirror.

### 3.6 Venting and cooling

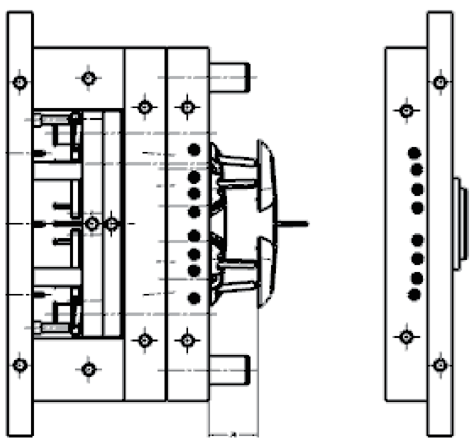
Trapped air in the mold needs to be pushed out by the injection pressure of the plastic material flow. To avoid fire caused by the pressure and temperature of the injected plastic, venting is required to drain the inside air. Venting is usually made on parting lines to free the trapped air out of the mold area.(8) However, we placed the vent on the insert core. Cooling is carried out as the coolant circulates the mold to cool the product and keep the temperature of the mold stable. For efficient venting and cooling, pressure and temperature sensors should be installed to estimate the exact time for cooling and minimize the chance of internal fire that causes product defects.

### 3.7 Lifter

A lifter in a mold is used to form the undercut of the product and mounted on an ejector plate (Fig. 8). The characteristics of the undercut are defined by the releasing direction, core-pulling force, and core-pulling distance.(14) The lifter was designed by geometric calculation that considers the angle and length of the lifter holder. The installation of the lifter demands considering the height of the undercut of the product and the angle of inclination of the lifter. The right triangle method is used to decide the position of the lifter. For the mold used in this study, the product stroke was calculated as 2.13 mm, and 3 mm was added for safety so that big enough product stroke parts can be ejected easily. The angle was 4.19°, which was rounded up to 6°.




(a)



(b)

Fig. 8. (Color online) Design of lifter of cover of motorcycle side mirror. (a) Mechanism of lifter. (b) Ejection mechanism of lifter.

## 4. Conclusions

To achieve the optimal design of injection molding for Industry 4.0, a newly designed injection molding process was proposed for the cover of a motorcycle side mirror. For mass production and according to the demand of the cover, the design needs snap-fit joints for the ease of manufacturing and use, and the lifter mechanism for the removal of the inner undercut of the product. The final design of injection molding was completed for the cover of a motorcycle side mirror that had two plates and was equipped with lifters as the ejection helper of the undercut. The opening for the mold was 250 mm, and the mold height was 630 mm. Multiple pressure and temperature sensors can be installed in the gate, runner, and cavity of the mold for manufacturing defectless products. The collected information can be transmitted to a molding machine for supplying the injecting material at a stable temperature and exerting an appropriate clamping force.

In the mold, the cycle time of the molding process was set to 20.5 s. The design of the injection molding was finalized for the molding machine (Toshiba IS220GN). Other machines can be used for molding with adjustments to meet their criteria. For efficient molding, the clamping force was calculated by using two different methods, and the results were 27.48 tons (Demag method) and 88.86 tons (Tat Ming method). The lifter was designed to demold the product from the mold as far as 70 mm and remove the undercut as far as 5.13 mm for safe ejection. The result shows the optimum design useful for other designs for solving similar problems of side mirrors of motorcycles.

## Acknowledgments

This work was supported by the Ministry of Science and Technology of the Republic of China under grant MOST 108-2628-E-029-001-MY3.

## References

1 C. Vatan: Prog. Rubber Plast. Recycl. Technol. (2003) 12. https://www.researchgate.net/publication/330105390_Recycling_of_plastic_material_in_design_world_examples_from_automotive_industry
2 J. C. Lin: J. Mater. Process. Tech. **120** (2002) 226. https://doi.org/10.1016/S0924-0136(01)01193-1
3 G. Menges, W. Michaeli, and P. Mohren: How to Make Injection Molds (Hanser, Munchen, 2001). https://doi.org/10.3139/9783446401808.fm
4 Y. Zhang and H. Zhou: Computer Modeling for Injection Molding (Wiley, New Jersey, 2013) p. 129. https://doi.org/https://doi.org/10.1002/9781118444887.ch5
5 M. W. Fu, A. Y. C. Nee, and J. Y. H. Fuh: Comput. Aided Design **34** (2002) 469. https://doi.org/10.1016/S0010-4485(01)00117-8
6 R. Karaman: Tetrahedron Lett. **49** (2002) 5998. https://doi.org/10.1016/j.tetlet.2008.07.171
7 B. Ozcelik, A. Ozbay, and E. Demirbas: Int. Commun. Heat Mass Transf. **37** (2010) 1359. https://doi.org/https://doi.org/10.1016/j.icheatmasstransfer.2010.07.001
8 S. Hwang, E. I. Reyes, K. Moon, R. C. Rumpf, and N. S. Kim: J. Electron. Mater. **44** (2015) 771. https://doi.org/10.1007/s11664-014-3425-6
9 H. Li and T. Tang: Appl. Mech. Mater. **37–38** (2010) 37. https://doi.org/10.4028/www.scientific.net/AMM.37-38.1507

10 M. Moayyedian, K. Abhary, and R. Marian: Procedia Manuf. **2** (2015) 241. https://doi.org/10.1016/j.promfg.2015.07.042
11 T. Ageyeva, S. Horvath, and J. G. Kovacs: Sensors **19** (2019) 3551. https://doi.org/10.3390/s19163551
12 C. Klahn, D. Singer, and M. Meboldt: Procedia CIRP **50** (2016) 264. https://doi.org/10.1016/j.procir.2016.04.130
13 J. S. Jang, J. W. Cho, M. S. Jeong, Y. J. Cho, S. K. Lee, W. J. Lee, J. W. Lee, and W. S. Yoo: Int. J. Precis. Eng. Manuf. **16** (2015) 1667. https://doi.org/10.1007/s12541-015-0219-6
14 B. Hou, J. Peng, S. He, Z. Huang, H. Zhou, and D. Li: Int. J. Adv. Manuf. Technol. **115** (2021) 329. https://doi.org/10.1007/s00170-021-06962-2


## About the Authors

**Chi-Hung Lo** is an associate professor in the Department of Industrial Design at Tunghai University, Taichung, Taiwan. His research interests include product aesthetics measurement, product family development, concurrent engineering in product design, and artificial-intelligence-based inference system design. (chlo@thu.edu.tw)

**Cheng-Chung Chen** is an associate professor in the Department of Hospitality Management at Tunghai University, Taiwan. His research focuses on computer-mediated communication (CMC), tourism marketing strategy, and food design. (jims@thu.edu.tw)

**Hwee Ling SIEK, Perline** is an assistant professor in De Institute of Creative Arts and Design, UCSI University, Kuala Lumpur, Malaysia. Her research interests focus on design education and cultural design. She is also an artist, she likes sketching and watercolour and acrylic painting.
(perlinesiek@ucsiuniversity.edu.my)

**Clara Metha Yoandara** is a master's degree student in the Department of Industrial Design at Tunghai University, Taichung, Taiwan. Her research interests include product family development and concurrent engineering in product design. (G09740701@thu.edu.tw)