International Journal of Research Publication and Reviews Vol (2) Issue (9) (2021) Page 448-455

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Design & Analysis of an Injection mould for SPRING SEAT of Strut Bearing by Reverse Engineering Approach

Abhimanyu K. [a] ****, L.G. Sannamani*** [b] ****, Dr. S.N. Ravi Shankar*** [c] *******

Student, M.Tech in Tool Engineering, GT&TC, Mysuru-16, INDIA.
HOD, Post Graduate Studies, GT&TC, Mysuru-16, INDIA.
Principal,Post Graduate Studies, GT&TC, Mysuru-16, INDIA.

A B S T R A C T

The aim of this work, titled "Design and Analysis of an Injection Mould for SPRING SEAT of Strut Bearing by Reverse Engineering Approach," is to design, analyze, and create a 3D model of a tool for a two-plate single cavity injection mould for the spring seat component of a strut bearing using a reverse engineering approach. The component is used in the strut bearing assembly.The material used to manufacture this component is Polyamide 66 is a glass-filled material with a 35 percent glass content. SOLIDWORKS software was used to create the tool's 2D designs and 3D model. The Easyfill advance of NX-CAD software was used to assess the gating position, weld line positions, air entrapment areas, and filling time, as well as to identify how the part fills. Tool design of the mould is included in this work.

Keywords:Injection Mould, Solid Works 2018, Submarine gate, PA 66 35% gf, NX Easy fill advanced, defects, tool design and assembly.

1. INTRODUCTION

Plastic products are made in a number of ways, from household convenience items to electronic devices and a wide range of other applications, including the world's strongest products, which are used in spacecraft, building projects, and other applications. The global use of plastic is projected to reach at least 18.2 trillion pounds (by weight). Extruders handle about 35% of the job, with injection moulding handling the remaining 33%. Blow moulding accounts for 11 percent, compression moulding for 4%, calendars for 5%, power form for 3%, coating for 6%, and other processes for 7% of the total. These proportions do not correspond to the number of machines in use. Injection moulding is one of the most often used techniques.

- High-speed manufacturing with minimal or no finishing required.
- Moulding complex shapes is easy.
- It's simple to keep tight dimensional tolerances.
- The entire procedure can be automated.
- There is less waste because the runners and gates can be reused.

By using a reverse engineering technique, this work addresses the designand analysis of a single-cavity two plate injection mould for the Spring seat of strut bearings. This component is utilized in strut bearings, which are manufactured of PA6.6 (Poly Amide 6.6) 35 percent GF (Glass Filled) material and are used in automobiles such as cars. In order to check the moulding parameters before moulding, the Easy-fill advanced program was used to do a mould flow study for the spring seat of a strut bearing component. Gating location, filling time, temperature at flow front, air traps, and weld lines are some of the elements to consider. The outcomes are analyzed. After analysis 3-D modelling of the tool has been done using SOLIDWORKS 2018 software.

* *Corresponding author.*
E-mail address: abhimanyuk296@gmail.com

2. OBJECTIVES

- The sample component is first 3D scanned, then the 3D scanned model is converted into a.stl file by completing the unfinished surfaces. After that, the.stl file is transformed to a STEP file. The entire procedure is carried out utilizing a reverse engineering approach.
- Finally, the STEP file is inspected for surface flaws, and any necessary revisions are made after consulting with the customer about tooling and functional needs.
- SOLIDWORKS 2018 is used to determine if a design part is mouldable by doing several analyses such as draft analysis, undercut analysis, thickness analysis, and selecting the optimal parting surface to determine the parting line.
- A study of selected materials was conducted to learn about their physical and mechanical properties as they relate to moulding material and features, both of which have an impact on tool design.
- The quality of the component is tested using mould flow analysis (Easy Fill Advance from NX-CAD) before it is manufactured, and the necessary corrections are made while building the tool.

3. METHODOLOGY

- The test component is 3D scanned in the first phase, and the 3D scanned model is then processed into STL format by completing the unfinished surfaces. After that, the STL file is translated to a STEP file.
- Surface defects on components are checked in the STEP file, and a mould model is created. The reference model and the workpiece are put together or manufactured.
- Gates and runners are incorporated as mould characteristics. They'll be considered when creating the moulded item, as well as when testing for interference throughout the mould opening process.
- Mould volumes are presently being removed in order to manufacture mould components. Once removed, the mould components are fully functional SOLIDWORKS parts that may be displayed in Part mode, used in Drawings, and exported.
- Easy-fill advanced of NX-CAD software was used to assess the gating position, weld line positions, air entrapment areas, and filling time, as well as to identify how the part fills.
- The mould's preliminary size is calculated, and a suitable mould foundation is chosen.
- The mould's base components have now been put together.
- The overall design, which includes the ejection system arrangement, is already complete.

4. THE PROCESS OF REVERSE ENGINEERING

The primary goal of RE technique is to create a conceptual model (example: triangulated surface) from a physical model, such as a sample (component or tool) or prototype. In this case, 3D scanning (digitizing) procedures with specific software for model rebuilding are required. The Flow Chart of 3d scanning is shown in Fig. 1

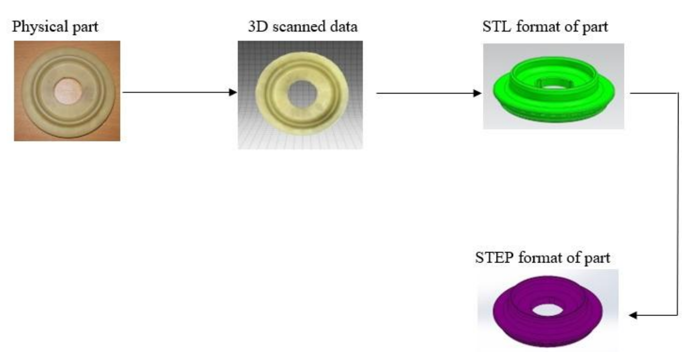


Fig. 1Stepsin 3D scanning (digitizing)

5. FEED SYSTEM CALCULATIONS

5.1 Sprue design

Standard sprue is selected by the designer and its diameter is modified according to the diameter of runner at one end and other diameter is derived from the relation,

$$D2 = D1 + 2L \tan A$$

Where,

D1= diameter of sprue at lower end = 4mm

D2=diameter of sprue at upper end

L= length of sprue selected by the designer = 70mm

A= tapered angle (2˚ to 5˚) =2˚ taken

D2= D1+ 2Ltan A

D2= 4+2(70)× tan (2˚)

D2 = 8.88 mm

Hence the diameter of sprue is 10 mm

5.2 Runner design

$$\textbf{Diameter of runner} = \frac{\sqrt[4]{L} \times \sqrt{w}}{3.7}$$

Where, w = weight of the component in g = 112.53 g

L = length of the runner = Radius of ID of component – Sprue radius – Gate distance

L = 16.52– 5– 4

L = 7.52 ≅ 8 mm

$$\text{Diameter of runner}(\emptyset) = \frac{\sqrt[4]{8} \times \sqrt{112.53}}{3.7}$$

Diameter of runner(∅) = 4.82 mm

For ease of machining and ease of cutter availability it is rounded off to **4 mm.**

5.3 Gate design

Diameter of gate

To find the diameter's arithmetical mean value. The average gate diameter d2 is calculated as follows:

$$\mathbf{d_2 = \frac{d_{2a} + d_{2b}}{2}},$$ Where d_{2a} is gate diameter.

d_{2a} =0.0042m +1.08

d_{2a} =0.0042m +1.08where m =112.53g

d_{2a} =0.0042(112.53) +1.08

d_{2a} = 1.55 mm

d_{2b} = ks

Where, s is inner wall thickness& k is coefficient which depends on wall thickness.

k = 0.5 - 0.8

The equations of coefficient k and gate diameter

d_{2b} were derived with reference to the equation of straight line

k = − 0.075 s + 0.8

k = - 0.075 s + 0.8 where s = 2.7 mm

k = - 0.075 (2.7) + 0.8

k = 0.5975

d_{2b} = k×s

d_{2b} = 0.5975 × 2.7

d_{2b} = 1.613 mm

$$d_2 = \frac{d_{2a} + d_{2b}}{2} = \frac{1.55 + 1.613}{2} = 1.58$$

d_2 = 1.58 mm

∴ Diameter of the gate is 1.58mm

Width of the gate

Width of the gate (Wg) is calculated by:

$$\mathbf{Wg = \frac{n \times \sqrt{A}}{30}}$$

Where n is the material constant, for Nylon n = 0.8

$$Wg = \frac{0.8 \times \sqrt{42509.84}}{30}$$

Wg = 5.5mm

6. MOULD FLOW ANALYSIS

EASYFILL ADVANCE was used to do a mould flow analysis on the injection mould. This is a distinct module in NX-CAD that one can use. The CAD-modelled part must be chosen and imported into the part adviser.

6.1 Processing Conditions of Pa66 35%Gf in mould Flow

Table 1 Processing Conditions of Pa66 35%Gf in mould Flow

Material Family	PA66
Material Grade	AKROMID A3 GF 35
Material Recommended Melt Temperature	300^0C
Material Recommended mould Temperature	90^0C
Maximum Injection Pressure	1810kg/cm^2

6.2 Procedure for mould flow analysis

- **Solid Modelling of Component:** NX CAD was used to create the component's solid model. Appropriate modeling methodologies were used to finish the model in accordance with the stated dimensions. The completed model is selected and loaded into the EASYFILL ADVANCE program.
- **Moulding Parameters:** Once the model has been transferred, the moulding settings must be input. Selecting a polymer from the adviser's current database or creating a new material by describing its properties are two of these possibilities. The most essential selection criteria are the provider's name and the brand name of the polymer on the market. One of the other moulding parameters is to enter the processing conditions. These include temperatures in the mould ranging from 80 to 100 degrees Celsius. It also necessitates entering the melt temperature range of 230 to 300 degrees Celsius. The other processing condition is the maximum injection pressure limit, which ranges from 10 to 500 MPa (101.97 to 5098.58kg/cm^2).
- **Positioning the Injection Point:** On the component or feed system to be checked, the injection point is set where it is most convenient. An imaginary boundary box around the component to be investigated can be drawn. The injection point's position relative to the bounding box can be modified in the required directions.
- **Analyzing:** After inputting the moulding parameters and choosing the gate placement, the part is available for analysis. The portion is filled with the required plastic material and processed as per the parameters during the analysis. The period of time it takes to analyze a part is influenced by the processing conditions as well as the component's geometric complexity.
- **Report Generation:** The results can be stored to a separate file and a report generated. The report contains a variety of results.

6.3 Best gate location

Once the material parameters such as Polymer, Material supplier, and Material trade name have been entered, it is important to know the Polymer injection/gate position to the cavity. The gate location result assigns a score to each point on the model based on its usefulness as an injection site. Given the material chosen, the result offers the optimal area to add an injection point.The areas that are most ideal for the injection location (colored red and rated as best) should be pursued as potential injection locations.

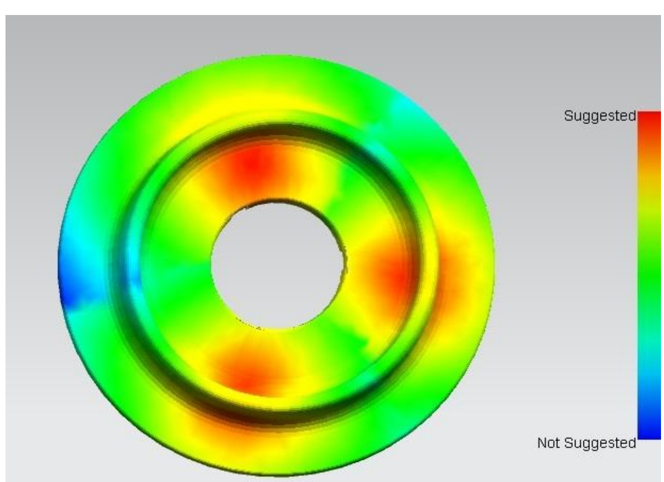


Fig. 2 Best gate locations

6.4 Feed system design and model

Submarine gate, U-shaped runner, and sprue are all part of the feed system. The submarine gate was chosen for a variety of reasons.

1. The submarine gate will leave a very modest gate impression on the component from an aesthetic standpoint.
2. During the mould opening, the submarine gate will automatically de-gate.

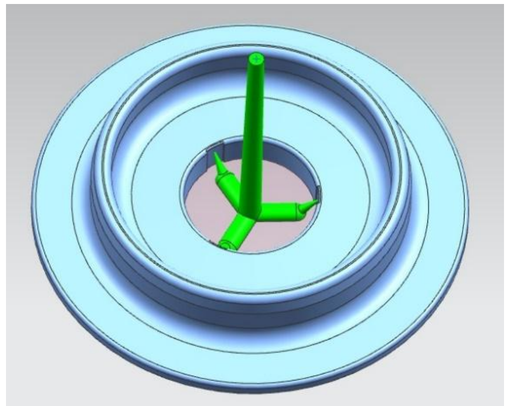
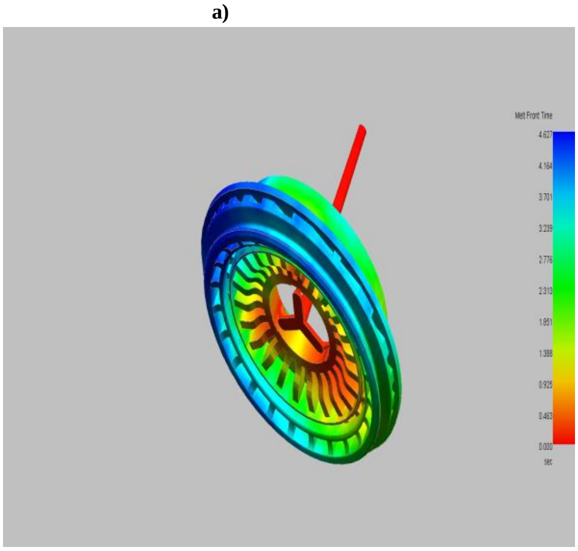


Fig. 3 Component with feed system

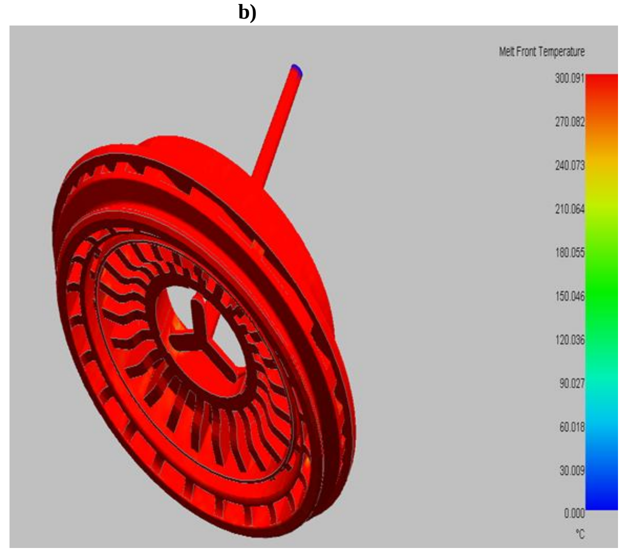
6.5 Analysis results

- The curve of fill time prediction is shown in Fig. 4(a). The time it takes to fill the impression is 4.26 seconds, according to the mould flow study. We can see from the plot that all of the component's extremities are filling at the same time, implying uniform filling of the part, which is critical to avoid differential cooling and obtain a high-quality moulded part. This also validates the gate dimensions.
- The flow front temperature is depicted in Fig.4(b). The melt temperature ranges from 300.09°C at the intake to 90.02°C at the end of fill, as shown in the graph. We may also estimate uniform filling of the section at the same temperature in most of the region based on the figure. Which is another essential aspect that has a significant impact on the product's quality.
- Figure 4(c) depicts the injection pressure needed to fill the cavity. The filling pressure is 6.24 MPa (63.63kg/cm^2), which is comfortably within the injection pressure capacity of the machine.
- The plot shows that the volumetric shrinkage is reduced, as indicated by the blue color range in Fig. 4 (d).
- The majority of air traps form along the parting line and may be easily vented out, according to the Air Traps finding. By separating the core insert near the rib, air traps on the central half of the rib can be prevented. Figure 4 (e) shows the blue dots that indicate air traps.
- Weld lines show at the bottom of the part, but there are none at the top. Weld lines will have no effect on the component's appearance. The weld lines in the component are shown in Fig. 4 (f).

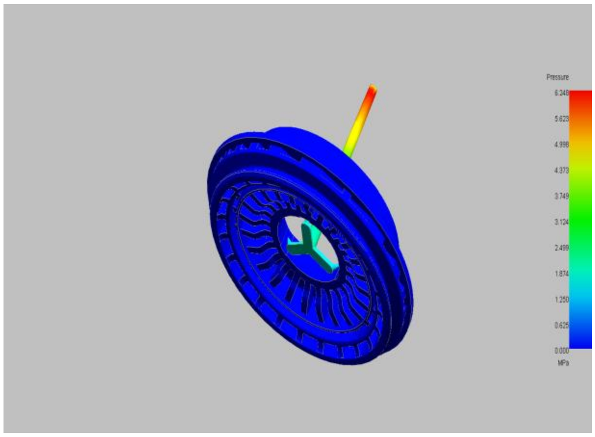
a)



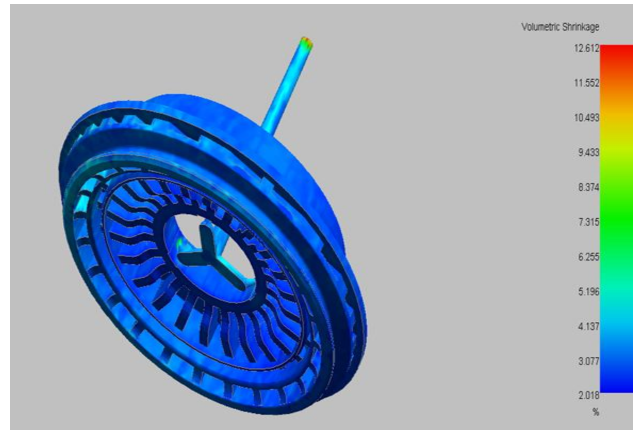
b)



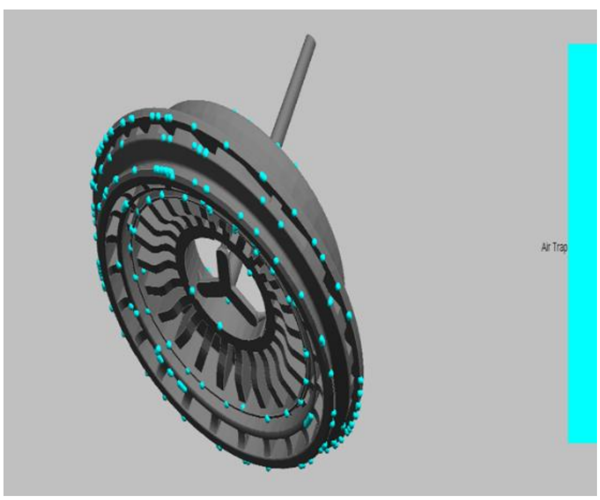
c)



d)



e)



f)

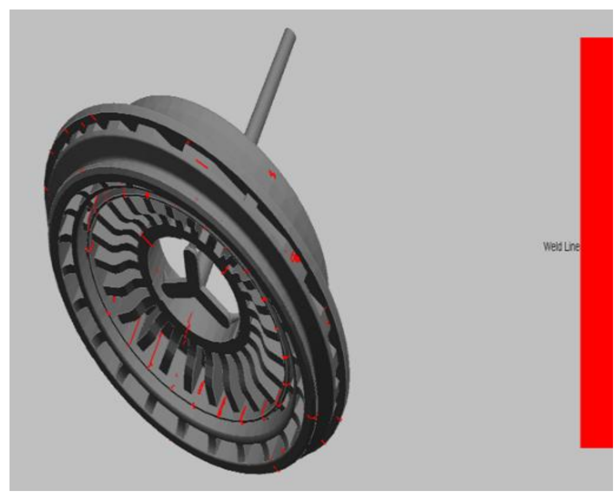


Fig. 4 Analysis results a) Melt front time b) Melt front temperature c) Pressure d) Volumetric shrinkage e) Air traps f) Weld lines

The design of the feed system is well suited for the moulding of the provided component, based on the above results of the mould flow adviser. All of the parameters were thoroughly examined, and the results were deemed to be good.

7. 3 D Mould Design

7.1 Assembly of fixed half.

The cavity inserts are fastened to the retaining plate via fasteners. The guide bush, sprue bush, cooling sleeves, and registration ring are all set in their respective locations. Finally, behind the top plate, the cavity housing plate is affixed. M12 screws, which penetrate through all of the plates and hold the fixed half assembly together, are used to secure the top plate. The assembly of fixed half is shown in the Fig. 5.

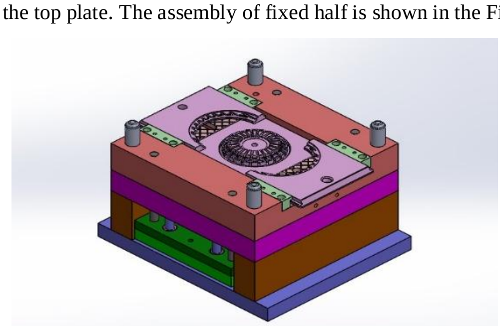


Fig.5 Top/ Fixed Half assembly

7.2 Assembly of moving half

The core insert(s) are fastened to the retaining plate with a fastener. The guide pillars are then secured to the core housing plate. The core rear plate is next placed and secured behind the core plate. Fasten the ejector back plate behind the ejector plate to complete the ejector mechanism. Ejector elements include things like ejector pins, return pins, and a sprue puller. The ejector directing pillars and bushes, as well as the support pillars, are all situated correctly. The ejector mechanism is free to move since it is positioned between the spacer blocks. The support pillars' and ejector mechanism's smooth movement is also ensured. To hold the bottom plate to the rest of the moving half assembly, large screws, such as M12 screws, are driven through the bottom plate, spacer blocks, core back plate, and core plate. The moving half assembly is depicted in Fig.6.

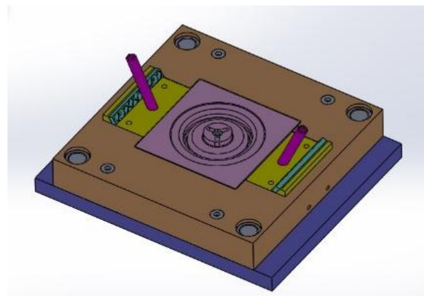
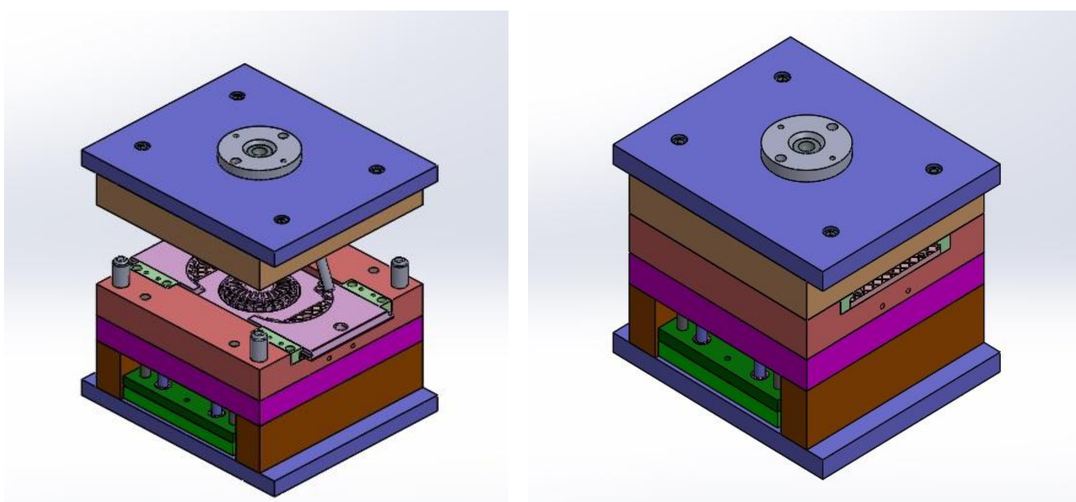


Fig. 6 Bottom/ moving half

7.3 Tool Assembly

After all of the mould elements have been designed, the entire mould assembly is examined, and then individual part drawings are prepared



a) b)

Fig. 7 a) Exploded view of two halves b) Tool assembly.

CONCLUSION

- Integrated reverse engineering is a novel approach for generating items in the research and development phase.
- Manufacturing time and associated costs for product development and management can be greatly reduced when this methodology is paired with modern rapid manufacturing technologies such as high-speed machining (HSM) and five-axis milling.
- The first stage in these production systems and procedures to record product geometry is 3D scanning (digitising).
- This research highlights some of the advantages and applications of reverse engineering methods and procedures in the manufacture of products and moulds, particularly when parts do not have 3D-CAD support.
- In this discipline, reverse engineering is crucial since it allows for the capture and digitization of item surface geometry for use in CAD/CAE/CAM.
- The application of reverse engineering in the development of quick plastic items is examined in this study. The project's purpose was to create a systematic design for an injection mould tool.

REFERENCES

[1] Sandeep Patil et al., Computer Integrated Injection Mould Split Design Reverse Engineering Approach, International Journal of Applied Engineering Research, Volume 6, Number 2011, pp. 1661-1672.

[2] Vikas B J et al., Influence of Feeding System in Injection Moulding for Lower Washer of a Bearing, IJRET: International Journal of Research in Engineering and Technology, Volume: 02, Issue: 08, Aug-2013.

[3] Sampath.A et al., Conceptual Design of Injection Mould tool for the Chair Hand Rest Component, IJRET: International Journal of Research in Engineering and Technology, Volume: 03, Special Issue: 03, May-2014.

[4] Haribabu Kummara et al., Design and Fabrication of Plastic Injection Moulding Tool for Pump Gaskets, International Journal of Advanced Engineering Research and Science (IJAERS), Vol-1, Issue-3, Aug- 2014.

[5] Vishwanath DC et al., Design of Singe Impression Injection Mould for Lower Bearing Cover, International Journal of Engineering Research & Technology (IJERT), Vol. 4, Issue 05, May-2015.

[6] Gurutheertha B. G et al., International Journal of Engineering Research & Technology (IJERT), Vol. 4, Issue 07, July-2015.

[7] Ingole Akshay Rambhau et al., Design and Analysis of Cover Handle Bar Top (Injection moulding process), International Journal of Current Engineering and Technology, MIT College of Engineering, Pune, India, AMET 2016, INPRESSCO IJCET Special Issue-4 , March 2016.

[8] R. Sedighi et al., Optimisation of gate location based on weld line in plastic injection moulding using computer-aided engineering, artificial neural network, and genetic algorithm, International Journal of Automotive and Mechanical Engineering, Volume 14, Issue 3, pp. 4419-4431, September 2017.

[9] Netaji Subhash Chandrashekhar et al., Simulation and Analysis of Step Light mid part using Mould flow analysis, International Research Journal of Engineering and Technology (IRJET), Volume: 06, Issue: 05, May 2019.