International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 08 | August - 2022 Impact Factor: 7.185 ISSN: 2582-3930

"Optimal Design and Analysis of Thermo-plastic product" (Chair Hand rest)

Sampath.A[1], Dr P.D.Sudersanan[2]

Mechanical Engineering, Sri Satya Sai University of Technology and Medical Science, Sehore[1],

Mechanical Department, Dr. T.Thimmaiah Institute of Technology, K.G.F [2]

sampath.a1987@gmail.com[1], dr.sudersanan@drttit.edu.in[2]

Abstract - The paper deals with the Design and mould flow analysis of injection moulding tool for Chair hand rest component. The part was analyzed with the mould flow software and accordingly, the injection moulding tool was designed. After mould flow analysis, considering the parameters (Possible gating location, runner location, fills time, Flow orientation, weld lines, etc) the manufacturing of the tool was done. Before the production of the component, there is a requirement of the moulding tool and moulding machine. The design is a very critical part and specialized area. Hence the tool should match the specification of the moulding machine to be accurate for producing the product.

Keywords- injection moulding tool, best gate location, fills time, weld lines, Air traps.

1. INTRODUCTION

Manufacturing process has made considerable progress over the century with the advent of new machines, high-performance cutting tools and modern manufacturing processes which enables today's industries to make parts faster and better than before.

Some of the products or components need to be produced in higher quantities with good surface finish, interchangeability, accuracy at lower costs. Such products are being produced in mass production using tools. The below mentioned are the few types of tools used for mass production: Press tools, Jigs and fixture, Die castings, Plastics moulds[7]

Now a day, the plastic products are replacing the metallic parts due to its advantages, like strength to weight ratio, corrosion resistance, aesthetics etc.

Plastic injection moulding process done by melting the plastic polymers and then it is been forced into a mould cavity. The solidification time depends on the size of the component. Once the plastic component is solidified, the part can be ejected. Injection moulded components are becoming the functional manufactured part in the modern world, from automotive products to food packaging. This versatile process allows us to produce high quality, simple or complex components. The injection mould tool is manufactured following the design procedure which is done by the Tool designers[1][2]

The moulding tools are designed depending on the clamping plate of the injection moulding machine used. Some of the factors considered while designing of injection moulding tool are [3]

1. Material of product
2. Mould flow analysis
3. Type of tool, Number of cavities
4. Component parting surface
5. Location of core and cavity
6. Ejection system
7. Designing of tool layout
8. Fool proofing arrangements
9. Cooling elements
10. Selection of Injection moulding machine

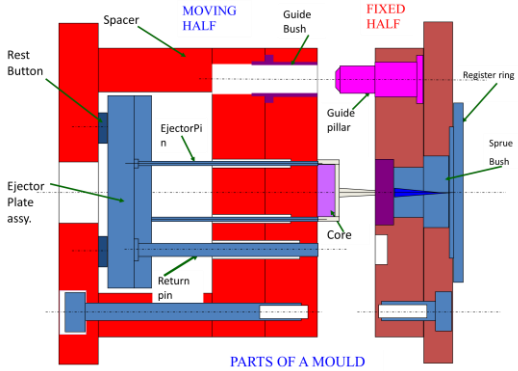


Fig-1: Injection Moulding tool

2. DESIGN METHODOLOGY

This paper highlights a practical design process and methodology of an injection moulding tool, die sets, adopted by analyzing the various parameters to produce the component chair hand rest [4]

Following are some details of methodology:

1. Detail study of the component.
2. Creating Solid modeling of the product.
3. Mould flow analysis of the component.
4. Creating the solid modeling of the tool to the desired values.
5. Selection of the different materials of each element in the tool and production to be carried out as per the design.
6. Tryouts and troubleshooting to be carried out before actual production.

3. INJECTION MOULDING CYCLE

The moulding cycle time starts from the moment when the mould closes, then the filling of cavity takes place and cooling cycle occurs, later the ejection of the component will be done when the mould tool opens after the cycle. The cycle time ranges depending on the size of component, approximately from 20 sec to60 sec.[4,5]

The moulding process basically involves 3 steps.

- Clamping of the tool
- Cavity filling
- Cooling and ejection

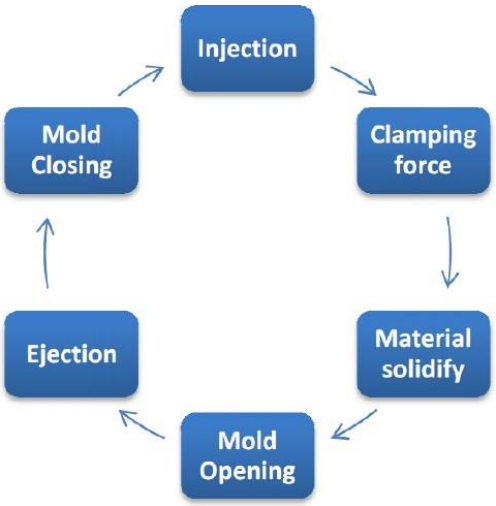
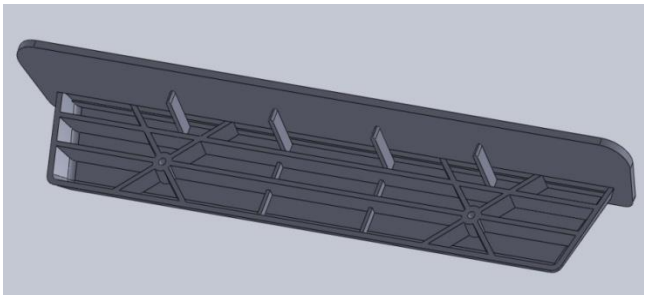


Fig-2: Injection Moulding Cycle

4. COMPONENT



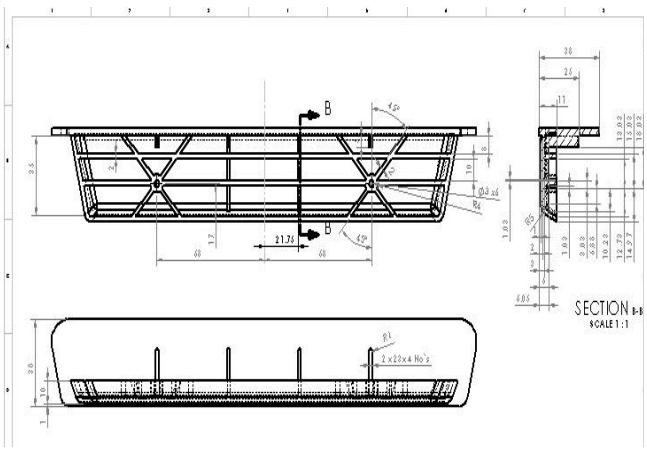


Fig-3: Component

Component details
Component name : Chair Hand Rest
Component material : PP(Polypropylene)
Shrinkage : 0.6 % - 1.2%
Weight of component: 54 grams
Moulding type : 2Cavity mould [6]

MOULDING TOOL ASSEMBLY

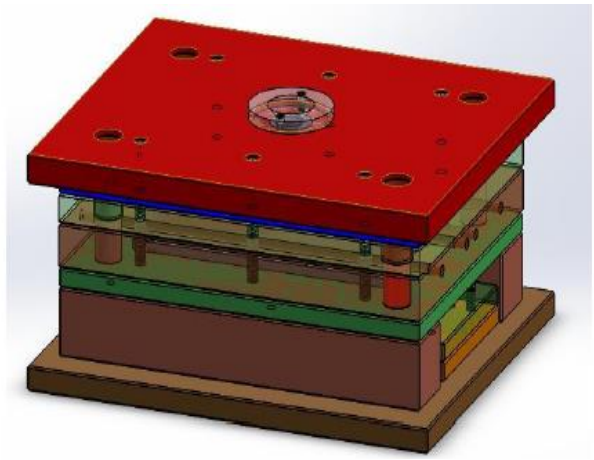


Fig-4: Moulding tool assembly

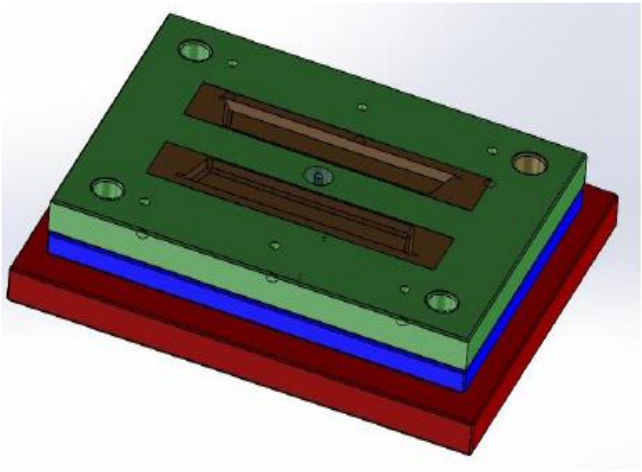


Fig-5: Fixed Half

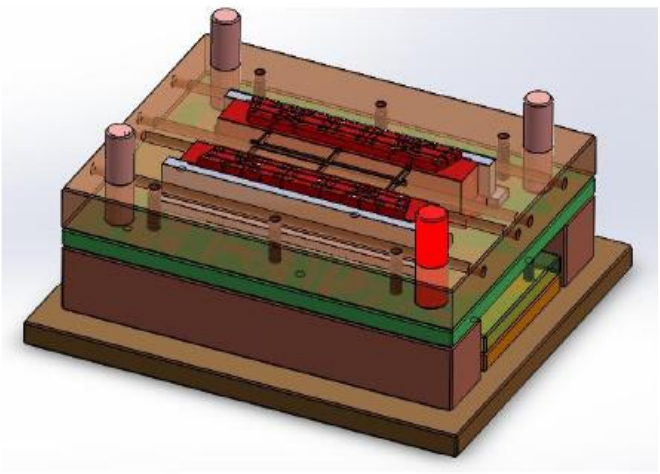


Fig-6: Movable half

5. MOULD FLOW ANALYSIS

The mould flow analysis is used to predict the probable defects occurring during the flow and solidification of molten plastic into the cavities. By this, injection moulding process parameters like injection time, mould temperature, (inputs) etc. can be optimized with the help of moulding window to obtain the desired quality (outputs like best gate location, fill time, warp, air traps, etc.) part/products.[2,4]

Steps involved in the mould flow analysis

- Import the 3D model of the component.
- Type of mesh
- Meshing and mesh correction.
- Runner, gate analysis to find the best gating location.
- Run the moulding window analysis to get the suitable parameters. i.e. mold temperature, melt temperature and injection time.
- Specify injection points.
- Specify the material used for component.
- Set processing parameters based on moulding window
- Set analysis for Fill + Pack.
- Run the analysis
- Interpretation of the results. [6]

Following are the results after analysis.

- Best gate location.
- Fill time.
- Flow temperature from the front.
- Air traps formation.
- The weld lines.
- Volumetric shrinkage.

Table-1: Data Input of the component

Material	Polypropylene (PP)
Shrinkage	1.4%
Density of material	0.9g/cm^3
Volume of component	60782.07mm^3(CAD Model)
Weight of the component	54.70 gms(CAD Model)
Melt Temperature	220ºC - 280ºC
Mould Temperature	30ºC - 80ºC

6. BEST GATING LOCATION

The position of the injection point on the component is important for filling. It represents the position where the plastic material to enter into the cavities. The best injection location creates a balanced flow in to the cavities. [8,9]

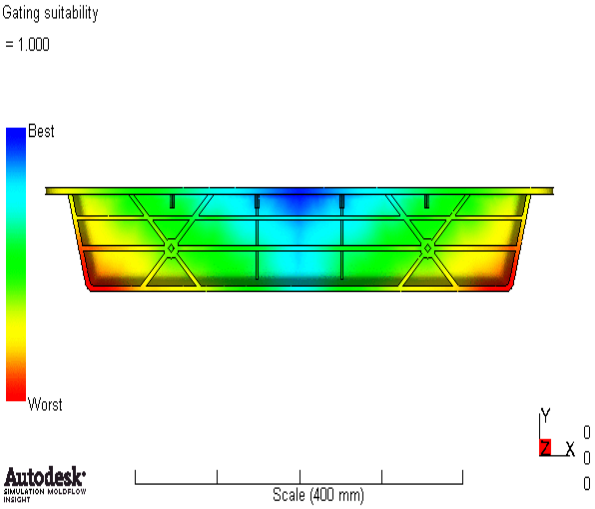


Fig-7: Gating location

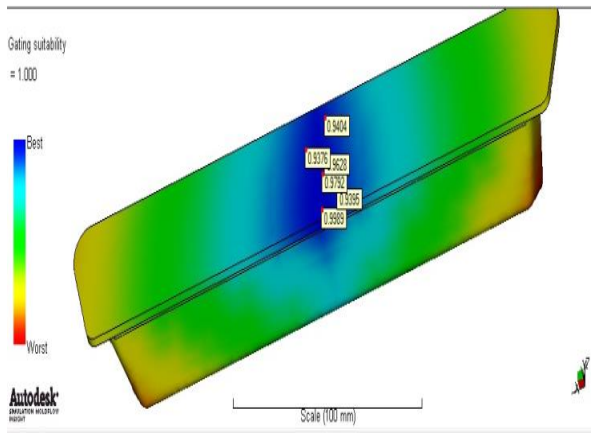


Fig-8: Best gating location

The blue colour portion indicates the best area for gate location and red colour indicates the undesirable location. From the above figure 8, we can see the best portion for gating location of the component which indicates 0.99, 0.98, 0.97 etc.

Due to the aesthetic point of view, the gate location has been shifted to the opposite direction of the component with three gates.

7. FILL TIME

Fill time is the time required to fill the molten material into the mould cavity. Time taken to fill the component is shown in fig 9. The red colour that is displayed indicates the last region to be filled and the blue indicate the first region to be filled within the specified time [8,9]. From the fig.9, it is concluded that there is no short shot in the component, hence filling will be completely done.

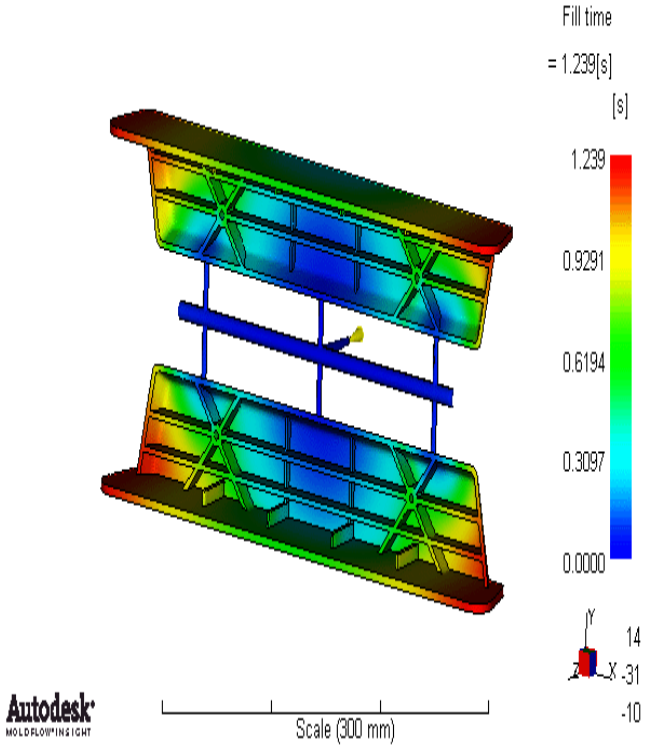


Fig-9: Fill time.

8. AIR TRAPS

Air traps occurs when air does not escape from the mould cavities. By this a bubble will be created inside the component, miss fill occurs and the pressure needed to fill the cavities will be more. The air traps can be seen in the Fig.10. The air traps were found more on the ribs of the component. So to avoid air traps, proper venting is incorporated in the mould near parting surfaces and ejection pin holes [8,9]. This is taken consideration in the tool manufacturing.

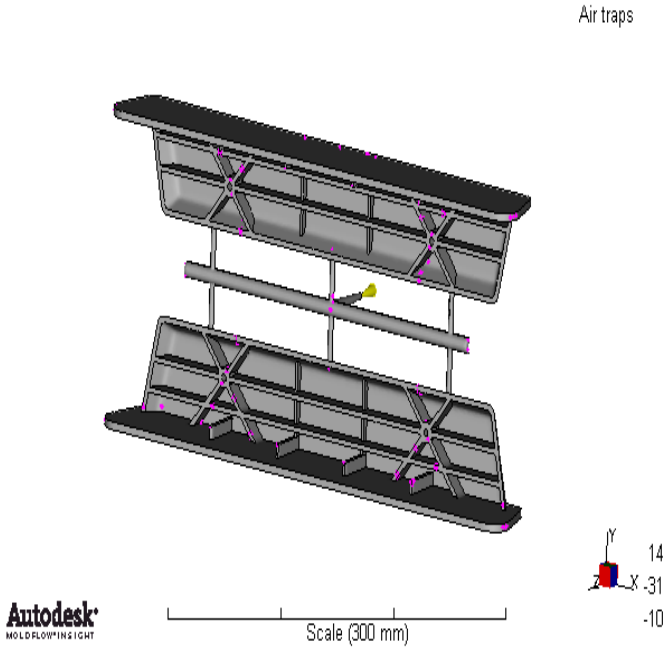


Fig-10: Air traps.

9. WELD LINES

Weld lines are the places where two flow melts have converged. The result indicates the presence and location of weld lines in the filled part model. Weld line on plastic part can cause the structural problems and be visually unacceptable. Therefore, weld lines should be avoided. It can be eliminated by maintaining uniform mould temperature[8].

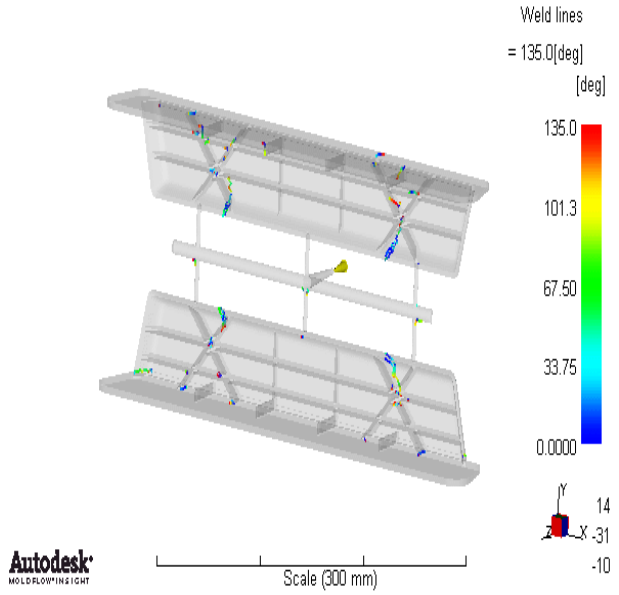


Fig-11: Weld lines

10. FLOW TEMPERATURE AT FRONT

The flow temperature from front is shown in figure 12.The molten liquid when flows towards the cavity, temperature changes on the component cross-sections. From the result, we can identify the different temperature inside the moulding [8,9].The temperature flow front range was found around 239.7ºC to 240.4ºC which is within the limit of the maximum melt temperature recommended by the supplier.

If the flow of the molten material temperature is too low from front of the feeding system, then short shot may occur. In areas where temperature is too high, material will flow but degradation and surface defects may occur. So, it should be properly analyzed and recommend for the temperature range of the polymer we are using.

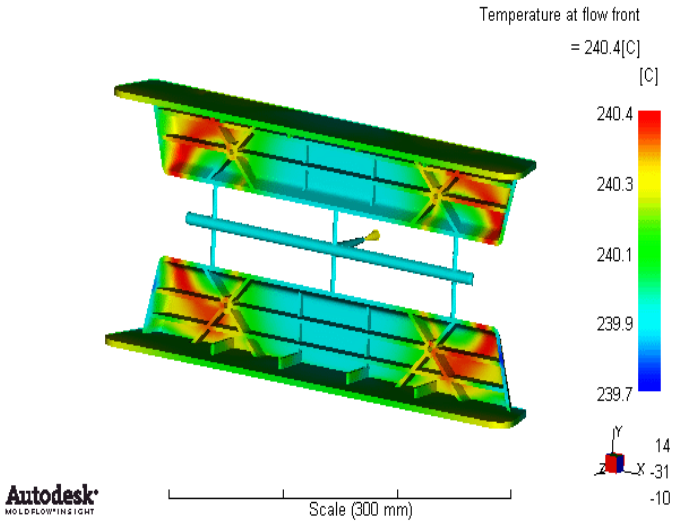


Fig-12: Temperature at Flow front.

11. CONCLUSION

1. From the analysis the best gating location is identified.
2. Due to aesthetic point of view, gating is provided on the opposite side of the component.
3. The temperature at flow front in the most region of the component varies from 239.7°C to 240.4°C.
4. Airtraps are more located on the rib surface, that can be removed by proper ventilation.
5. Most of the weld lines are located on the ribs, which can be eliminated by maintaining uniform mould temperature.

.

12. REFERENCES

[1] T. Mohan Kumar at al., "Optimization of Process Parameter In Injection Moulding Using Taguchi Method", International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 06 Issue: 05 | May 2019.

[2] **P.K. Bharti, M. I. Khan,** Recent methods for optimization of plastic injection molding- a retrospective and literature review. International Journal of Engineering ScienceandTechnology Vol.2(9),2017,4540-4554

[3] Haribabu Kummara and Dr. G. Harinath Gowd, "Design and Fabrication of Plastic Injection Molding Tool for Pump Gaskets", International Journal of Advanced Engineering Research and Science (IJAERS) [Vol-1, Issue-3, Aug- 2014] ISSN: 2349-6495.

[4] Dr. M P Singh and Akhilesh Jaguri , "Gating System Design Optimization for Injection Moulding", International Journal of Mechanical and Industrial Technology Vol. 3, Issue 1, pp: (156-163), Month: April 2015 - September 2015.

[5] RashiA.Yadav et al., "Recent Methods for Optimization of Plastic Injection Molding Process - A Literature Review", International Journal of Scientific & Engineering Research Volume 3, Issue 12, December-2012 ISSN 2229-5518.

[6] Mahendrasinh M. Raj, Hemul V. Patel, Lata M. Raj and Naynika K. Patel, "Studies on mechanical properties of recycled polypropylene blended with virgin polypropylene," international journal of science invention today, Volume 2, issue 4, pp. 194-203, (2013)

[7] Pratibha V Patil et al., "Design and Fabrication of Mini Injection Molding Machine", IJRASET Volume 5 Issue IV, April 2017 IC Value: 45.98 ISSN: 2321-9653.

[8]Technical Directory on Design and Tooling for Plastics, CIPET

[9] Injection Mould Design, R.G.W. Pye