International Journal of Research Publication and Reviews

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

Design and Simulation Study of Mixer Jar Lid Component using NX-11.0 and Autodesk Moldflow Adviser Software

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

[a]*Student, M.Tech in ToolEngineering,. GT&TC, Mysuru-16,karnataka, INDIA*
[b]*HOD, Post Graduate Studies, GT&TC, Mysuru-16, Karanatka, INDIA*
[c]*Principal, Post Graduate Studies, GT&TC, Mysuru-16, Karanatka, INDIA*

A B S T R A C T

In the earlier days of injection moulding,mould making industries were following difficult procedures and was very challenging. Andentire design were to be completed by drafting done by manual drawings. As the drafting and modeling softwaresstarted entering into the market, designing was carried out models with the help of 2-d mould, but these designs were never close to real mould. Later onin modern day mould designs, with the introduction of 3-D mould design softwares, designing became lot more easier. In the present investigation, design of feed system was carried out using NX-11.0 software. And with the help of analysis onecan predict the plastic behavior which happens in the mould.These analysis softwares brought down computational time by a great margin. This paper explains the design and analysis study of the component of a mixer jar lid. With the help of Autodesk Moldflow Adviser software, analysis of component was made and results are used for the designing of mould for the same component. The material selected in this study is polycarbonate.When defects were found, the feed system was modified and the analysis was carried out until flaw-less output was obtained. The defect free output which leads to excellent quality of the productwas followed by final tool design.

Keywords: Injection Mould, NX-11.0, Rectangular edge gate, Polycarbonate, Analysis, Autodesk software, defects, tool design and assembly.

1. Introduction

Plastic is widely used everywhere in the world because of its desirable properties. It can replace the metal with strength and drastic reduction in the weight. Polycarbonates are the thermoplastics of high-grade polymers. This material can be easily mouldable. It has high strength and transparent. The major drawback of this material is, it isn't flexible and not applicable for aesthetic colour component.

Injection moulding for plastic component is common method in the plastics processing industry. It is the process that involves the cyclic operations of fast filling followed by cooling and ejecting the component. Selecting the proper type of mould is must for producing the component and it is very crucial because it has the direct impact on product quality, speed of production, and overall costs.

In the present work component mixer jar lid is the component that needs to be analyzed in order to get proper ideology of the mould. The analysis helps to understand the plastic flow behavior inside the mould. The analysis software like Autodesk Moldflow Adviser helps in achieving the analysis of components. This paper aims to find the proper parameters required for making the mould. The required basic calculations were made and considered those parameters in the analysis.

Analysis was be carried out to know the proper filling, weld lines, entrapped air, injection time, temperature and pressure distribution in the component. The component Mixer Jar lid is considered for the analysis and results were used in the modelling of the mould in NX-11.0 software.

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

2. Objectives

- The primary goal of this project is to designand analyse a three-dimensional mould tool.
- For creating a simple and viable mould, a basic component study must be completed.
- Autodesk Moldflow Adviser is used to check component quality prior to mould production.
- Using analysis results, correcting any potential problems or defects that have arisen.

3. Methodology

3.1 Study of the component: This step involves a thorough examination of the component that was provided by the customer. Checking the component's dimensions, whether it's moldable or not.

3.2 Feasibility study of the component: This stage is crucial in the design and fabrication of the mould. Because this process involves a thorough examination of the component for design purposes, such as parting line details, core and cavity draft information, any changes to the part are addressed with the client, and any necessary corrections or alterations were made.

3.3 The design concept is being prepared: Finding out the mechanisms, part ejection, gating type and location, a preliminary hand design of the layout that has the least essential features, and cooling for the component are all basic needs for the making the mould.

3.4 The component analysis: The component will be analysed to determine the component entire filling, weld lines, air traps, injection time, temperature, and pressure distribution. These considerations will aid in the creation of a mould that produces high-quality items.

3.5 Design of a three-dimensional mould: Using the techniques outlined above, creating a 3-D mould using NX-11.0 software. The 3-D mould will be utilized to create the mould, as well as to assemble and produce the component.

4. Study of the component

Table 1. Component Details

Name of the component	Mixer jar lid
Average thickness	2 mm
Material	POLYCARBONATE (TRANSPERANT)
Shrinkage	0.5-0.8 %
Density	1.2 – 1.5 g/cc
L× W × H (mm)	167× 117 ×44
Volume (v)	58 cm^3
Melt temperature	288 – 316 ℃
Mould Temperature	71.1 – 93.3 ℃

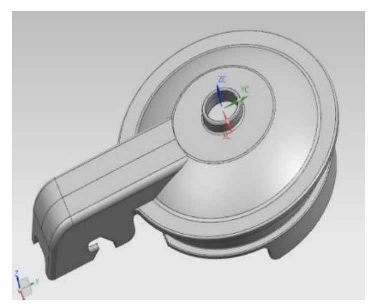


Fig. 1. Component Image

5. Feasibility study:

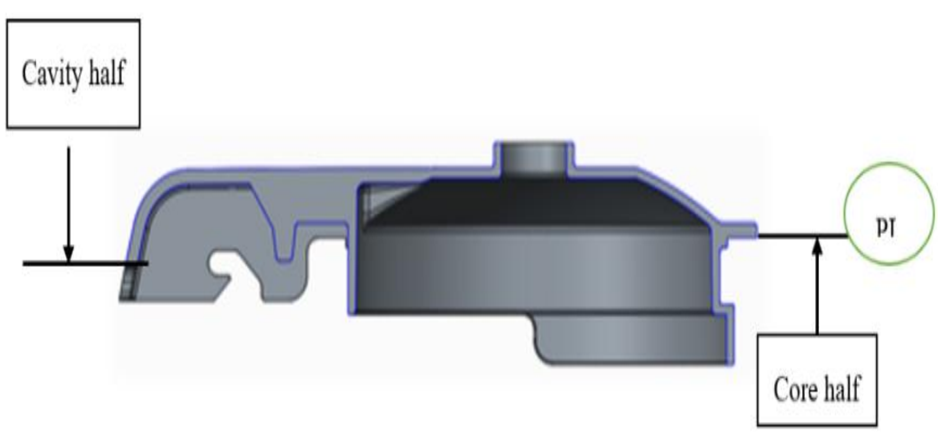


Fig. 2. Parting line representation

The geometry and structure of the component influence the parting surface selection. This facilitates the manufacture and ejection of the component. The underlined surface was used as the parting surface in the 3D model

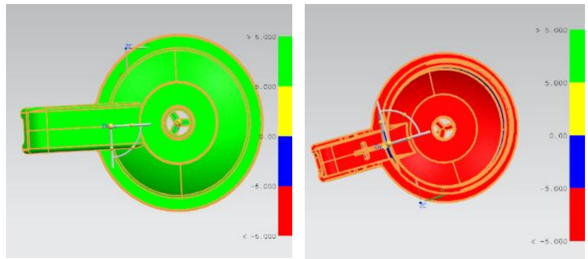


Fig. 3 Draft analysis

Draft Analysis allows oneself to determine if the drafted element can be easily removed. At every stage in this form of study, colour ranges define zones on the examined element where deviations from the draft path correspond to specified values.Here green colour indicates positive drafts and red colour indicates negative drafts. Ejection of the component was made on negative drafting.

6. Design of feed system

Runner design: Diameter of runner can be calculated by using the relationships

$$Rd = \frac{\sqrt{Wt} \times \sqrt[4]{L}}{3.7}$$

Shot weight of the component is 73.08 g

Length of the Runner is 16 mm

$$Rd = \frac{\sqrt{73.08} \times \sqrt[4]{16}}{3.7} = 4.62 \approx 5 \text{ mm}$$

For ease of machining and ease of cutter availability it is rounded off to 5 mm.

Gate design: The Rectangular edge gate is considered for the design purpose and dimensions are given below.

Width of the rectangle W = 4 mm

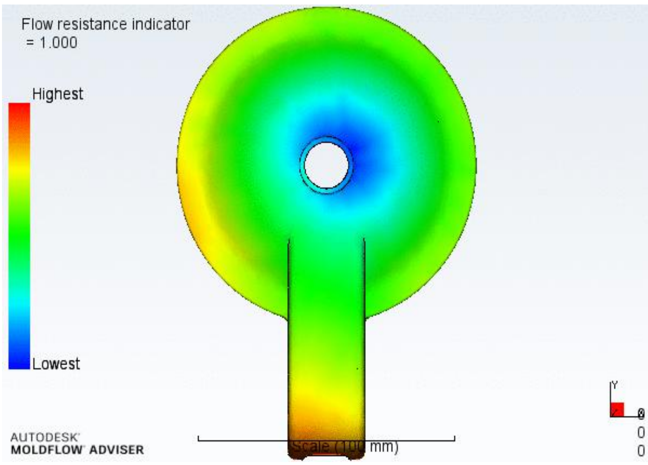
Depth of the rectangle H = 1 mm

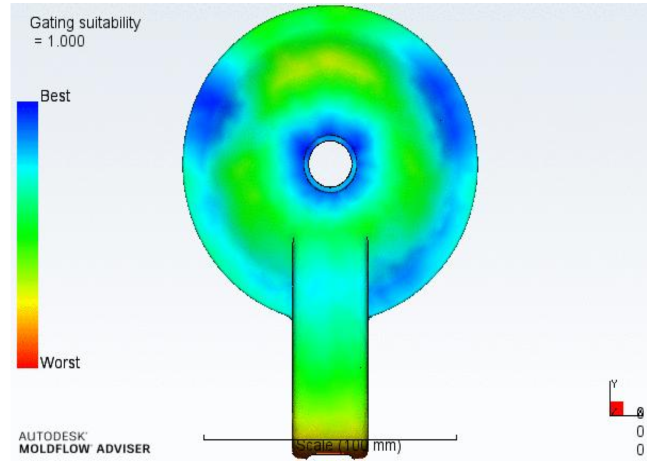
Thickness of the rectangle t = 1 mm

7. Flow Analysis results

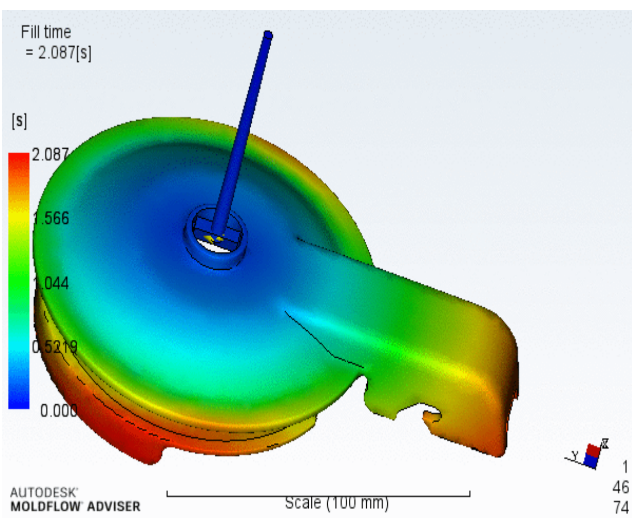
Autodesk Moldflow Adviser software was used to perform the mould flow analysis. The software can predict fill time, injection pressure, air traps, and possible shrink marks based on selected gate and material properties.

- The region where plastic flow will be easy and difficult is depicted in Fig. 4 (a). The red region has the highest resistance, while the blue sector has the easiest flow.
- The optimum gating site where the gate can be created is shown in Fig.4 (b). Because the mould is single cavity, the core region has the best moulding advantages.
- Figure 4 (c) depicts the component's actual filling time. It takes 2.087 seconds to fill the component completely.
- The adviser is confident in the component's filling. It's also displayed in Fig. 4 (d) with a 100% high confidence fill.
- The advisor predicts the pressure needed to inject the material into the mould spaceas 119.9 MPa as shown in fig.4 (e).
- Shrinkage is the deformation of a plastic moulded component caused by a density difference between melt and solid plastic. When the plastic cools after the injection, it begins to shrink. Proportion of shrinkage is nearly 5%, as shown in Fig. 4(f). It's also acceptable from a design perspective.
- Sink marks are tiny pits or sorrows that form in the injection moulded part's thicker spaces.The possible sink areas are depicted in Fig. 4(g) The maximum depth of a sink is 0.05 mm, while the average is 0.02 mm. 3.22 percent of models are sensitive to shrinkage, which is acceptable.
- Warpage occurs when a component is distorted during the cooling process. It has the potential to cause the part to fold, bend, or twist. There is no indication of warpage based on Fig. 4(h).
- Air traps are most commonly found in the last fill sections. The absence of undersized design and vents is a common source of air traps.Figure 4 (i) depicts air trapped in the outer region, which can be removed by constructing adequate vent routes.
- Figure 4 (j) depicts the component's available weld linesand has the lowest weld lines.

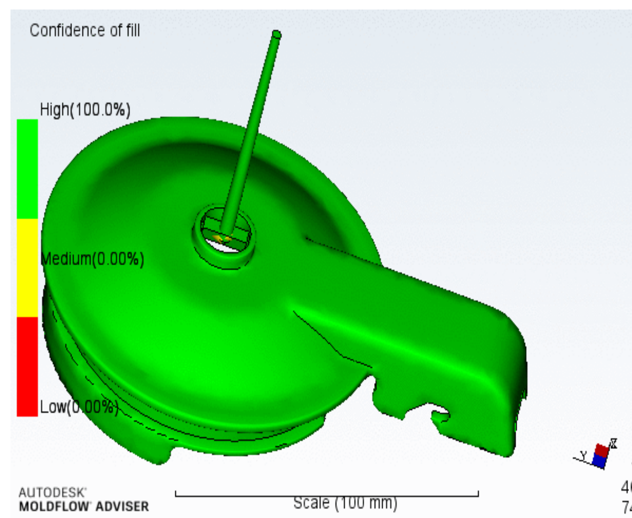

Flow resistance indicator
= 1.000
Highest
Lowest
AUTODESK
MOLDFLOW ADVISER
Scale (100 mm)


Gating suitability
= 1.000
Best
Worst
AUTODESK
MOLDFLOW ADVISER
Scale (100 mm)

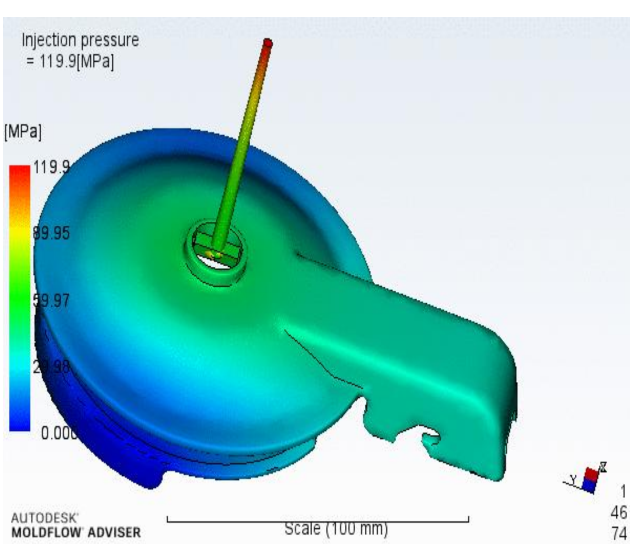
a) b)


Fill time
= 2.087[s]
[s]
2.087
AUTODESK
MOLDFLOW ADVISER
Scale (100 mm)

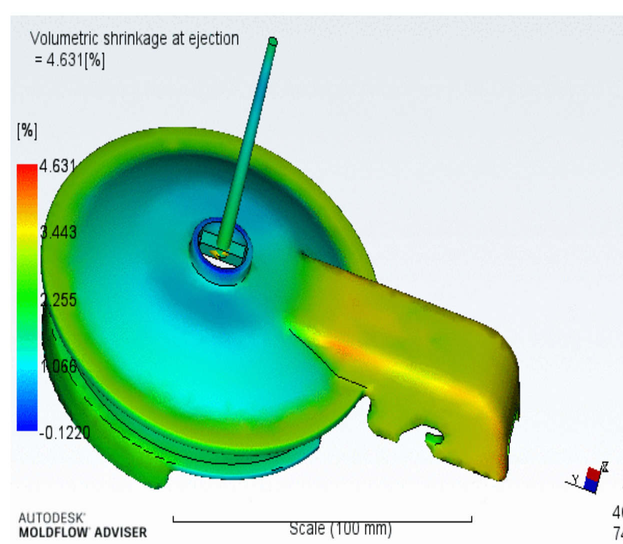
c)


Confidence of fill
High(100.0%)
Medium(0.00%)
Low(0.00%)
AUTODESK
MOLDFLOW ADVISER
Scale (100 mm)

d)

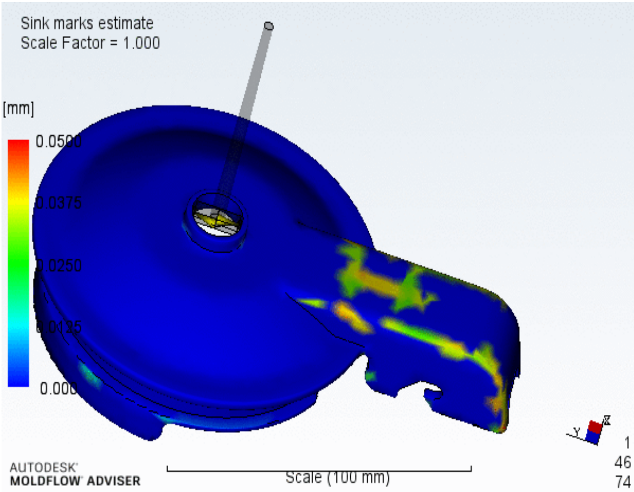

Injection pressure
= 119.9[MPa]
[MPa]
AUTODESK
MOLDFLOW ADVISER
Scale (100 mm)

e)

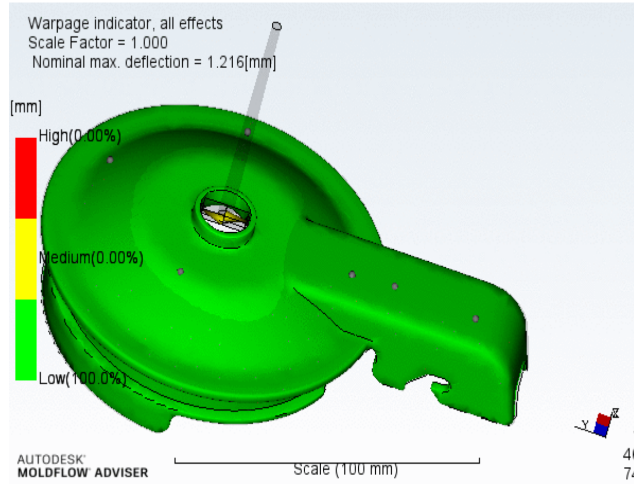

Volumetric shrinkage at ejection
= 4.631[%]
[%]
AUTODESK
MOLDFLOW ADVISER
Scale (100 mm)

f)

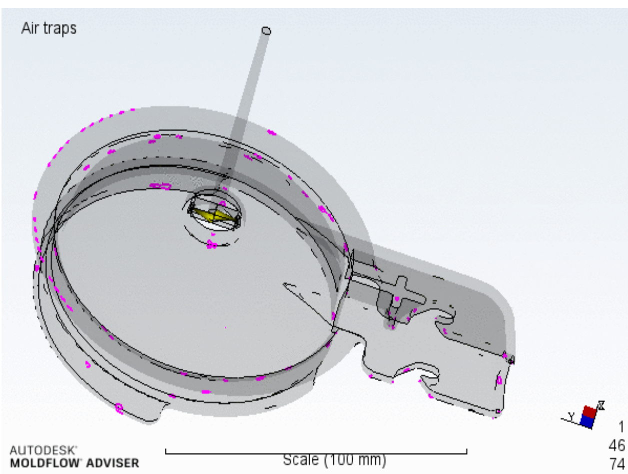
International Journal of Research Publication and Reviews Vol (2) Issue (8) (2021) Page 1538-1544



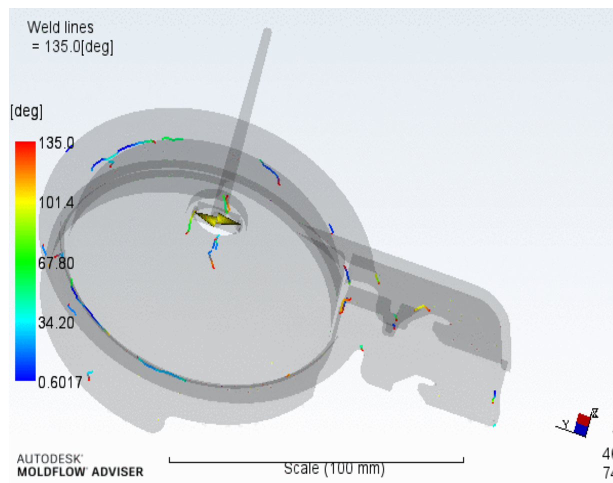
g)



h)



i)



j)

Fig. 4 a) Flow resistance indicator b) Gating suitability
c) Fill time d) Confidence of fill
e) Injection Pressure f) Volumetric shrinkage
g) Sink marks estimation h) warpage indicator
i) Air traps j) Weld lines

8. Design of a three-dimensional mould

8.1 Shrinkage addition

Plastic shrinkage is a dimensional change in the moulded object that occurs when it cools following injection. The majority of the shrinkage happens while the component is still in the moulding tool during the cooling stage, however the part may shrink by a minor percentage after ejection as it cools.Before initiating the core and cavity extraction, shrinkage should be applied to the component to eliminate the shrinkage impact caused by the polymer's density reducing. 1+s is the formula used.

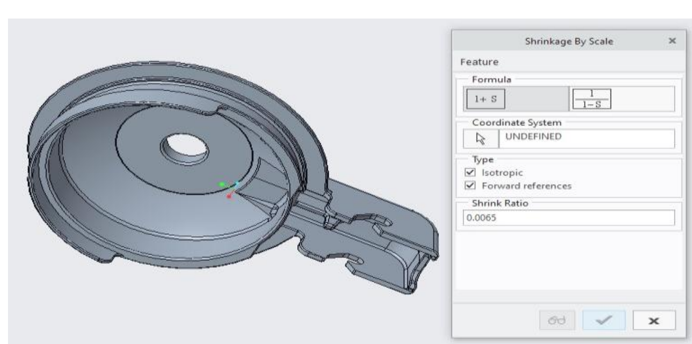
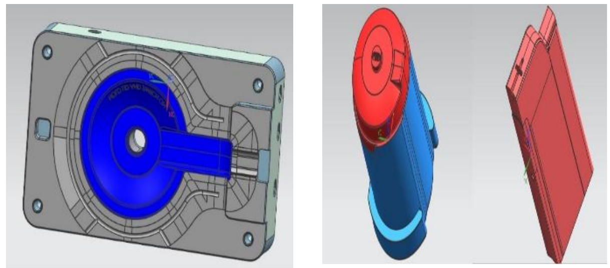


Fig. 5 Adding Shrinkage

8.2 Core and Cavity inserts



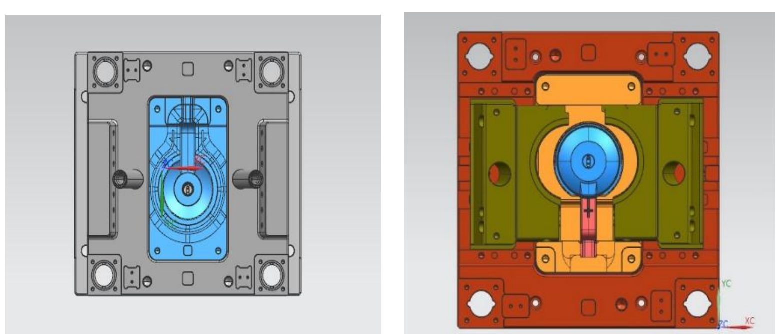
a) b)

Fig. 6 a) Cavity insert b) core inserts

The areas of core and cavity surfaces are present in core and cavity inserts. Because these inserts come into close contact with melt injected material, they must be composed of a tough substance. D2-HP4MA material is used for the insert high harden steel material.The cavity insert is depicted in Fig. 7.2, with the highlighted area representing the cavity of the component.Tiny steel blocks are used to make the inserts. Core inserts are the male component of the inserts. The inserts are secured in the bolster or housing with screws and fittings.

8.3 Two halves

A standard two-plate mould has one fixed half and one moving half. All of the mould components are either assembled in the fixed half or the moveable half. The separating surface is the line surface where these two halves meet. The fixed half of the mould is attached to the machine's fixed platen, while the moving half is connected to the machine's moving platen.In most cases, the core is located in the moving half of the mould, while the cavity is located in the stationary half. The ejection system is also included in the mould's movable half. This is due to the fact that the portion has shrunk. After the part shrinks while cooling, the core sections of the component remain when the mold is opened.The elements for the feed system are all provided in the cavity half of the mold. Because the fixed platen and half are where the plastic material is injected, parts like the registration ring and sprue bush should only be housed in the fixed half.



a) b)

Fig. 7 a) Top half assembly b) Bottom half assembly

8.4 Tool Assembly

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

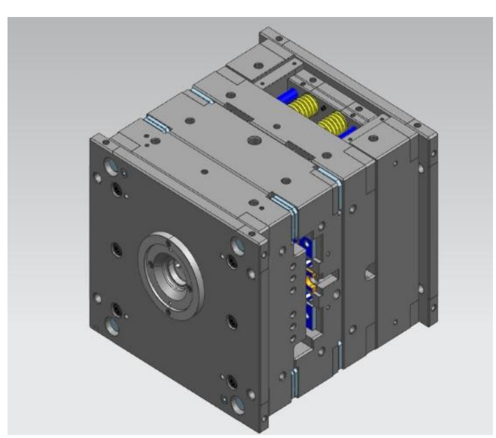


Fig.8 Assembly of the tool

9 Conclusion

- This work explains the creation of a plastic injection mold for producing Mixer Jar Lid.
- A study of the component material was conducted to assess its mechanical, thermal, and electrical qualities.
- Calculations are done to determine the machine's suitability for mould manufacture.
- Moldflow Advisor software is used to perform and understand the analysis. Before attempting to rectify them, a thorough examination of the types of flaws and the underlying causes of the problems is required, as is the construction of a tool that appears to perform satisfactorily during testing phases and is assembled after following the manufacturing procedure.

REFERENCES

[i] Mr. M. G. Rathi, Mr. Manoj DamodharSalunke, "Analysis of Injection Moulding Process Parameters". IJERT, Vol.1 Issue 8, October – 2012.

[ii] N. Sreenivasulu, Dr. D. Ravikanth, "Injection Moulding Tool Design Manufacturing, Estimation and Comparison of L&T Power Box Side Panel Using Plastic Materials HDPE, ABS, PP and PC". IOSR-JMCE, Vol.8, Issue 3 October – 2013.

[iii] Kiran Tom Thomas, Ramesh Babu K., "Conceptual design of two plate injection mould tool for five pin Daimler regulator". IJMERR, Vol.3 Issue 4, October – 2014.

[iv] Ajith Holla U., Ramesha N., "Determination of Process Settings for Injection Moulding Tool by use of CAE Software Tools". IJERT, Vol.4 Issue 5, May – 2015.

[v] Shaikh Mohammed Ishak, Shaikh Mohammad Nabil, Sheliya Mujahid Ilyas., "A Simulation Study of Gates and Cooling Systems in Plastic Injection Molding". IJERT, Vol.7 Issue 3, March – 2018.

[vi] V. V. Gaddamwar, Dr. S. P. Bhosle, "Optimization of Process Parameter for ABS, Nylon, Polypropylene Material in Injection Molding using Taguchi Method". IJERT, Vol.8 Issue 6, June – 2019.

[vii] Prashant K. Mulge, Dr. Shashidhar S. kalashetty, "A Brief Literature Review on Optimization of Plastic Injection Moulding Process Parameters for Various Plastic Materials by using Taguchi's Technique". IJERT, Vol.8 Issue 7, July – 2019.

[viii] A. Glanvill, E. Denton, "Injection mould design fundamentals". Industrial press, 1965.

[ix] R.G.W. Pye, "Injection mould design". Affiliated East-West Press Pvt Ltd, 4th edition,2000.

[x] Dominick V. Rosato, Donald V. Rosato, Matthew V. Rosato, "Plastic Product Material and Process Selection Handbook". September 2004.

[xi] "Arburg Practical Guide to Injection Moulding". Edited by V. Goodship. 2004.

[xii] Gastrow, "Injection molds 130 proven designs". 4th edition. 2006.