

**CZECH TECHNICAL UNIVERSITY IN PRAGUE**

**FACULTY OF MECHANICAL ENGINEERING**



Injection of test specimen for tensile test

Bemmireddy, Santhosh Reddy

2019

Supervisor: Ing. Jiri Moravec, Ph.D.



# MASTER'S THESIS ASSIGNMENT

## I. Personal and study details

Student's name: **Bemmireddy Santhosh Reddy** Personal ID number: **473334**  
Faculty / Institute: **Faculty of Mechanical Engineering**  
Department / Institute: **Department of Process Engineering**  
Study program: **Mechanical Engineering**  
Branch of study: **Process Engineering**

## II. Master's thesis details

Master's thesis title in English:

**Injection of test specimen for tensile test**

Master's thesis title in Czech:

**Všříkování zkušebního tělesa pro tahovou zkoušku**

Guidelines:

Design the injection molding process parameters for injecting a test specimen for a tensile test. The specimen will be made from chosen thermoplastic material. The design can be done using any simulation software. Try to verify the proposed parameters on the laboratory injection molding machine.

Bibliography / sources:

according to supervisor's recommendation

Name and workplace of master's thesis supervisor:


**Ing. Jiří Moravec, Ph.D., Department of Process Engineering, FME**

Name and workplace of second master's thesis supervisor or consultant:

Date of master's thesis assignment: **28.04.2019** Deadline for master's thesis submission: **07.06.2019**

Assignment valid until: \_\_\_\_\_

  
Ing. Jiří Moravec, Ph.D.  
Supervisor's signature

  
prof. Ing. Tomáš Jirout, Ph.D.  
Head of department's signature

  
prof. Ing. Michael Valášek, DrSc.  
Dean's signature

## III. Assignment receipt

The student acknowledges that the master's thesis is an individual work. The student must produce his thesis without the assistance of others, with the exception of provided consultations. Within the master's thesis, the author must state the names of consultants and include a list of references.

**30-04-2019**  
Date of assignment receipt

  
Student's signature

# Annotationsheet

**Name:** Santhosh Reddy

**Surname:** Bemmireddy,

**Title Czech:**

**Title English:** Injection of test specimen for tensile test.

**Scope of work:**Number of pages: 53

Number of figures: 35

**Academic year:** 2017 - 2019

**Language:** English

**Department:** Faculty of Mechanical Engineering

**Specialization:** Process Engineering

**Supervisor:** Ing. Jiri Moravec, Ph.D.

**Reviewer:**

**Tutor:**

**Submitter:**

**Annotation - English:** This master thesis deals with the literature survey on injection molding machine construction, working and process parameters. The main aim of this thesis is to perform simulations and compare them with machine settings to understand the principle parameters of the injection molding machine by both theoretically and practically for the study of improving product quality. This involves comparing the Practical results of the parameters of the injection molding machine and the suggested results by simulation from the software. The change in parameters will give the better quality of the product.

**Keywords:** Injection moulding machine, processing parameters of injection moulding machine, working principle, Plastic flow analysis, Quality of the product, simulation, analyzing the real processing data of the machine.

**Utilization:** For Department of Process engineering, Czech technical university in Prague.

**Declaration Statement**

*I declare that I have worked out this thesis independently assuming that the results of the thesis can also be used at the discretion of the supervisor of the thesis as its co-author. I also agree with the potential publication of the results of the thesis or of its substantial part, provided I will be listed as the co-author.*

In Prague .....09/08/2019.....Signature.....*B. Benoit*.....

## **ACKNOWLEDGEMENT**

First, I would like to thank god for all things which I have in life, and I would like to thank my teachers and colleagues who are all helpful in so many ways in completion of this thesis and my master program. And I would like to express my sincere gratitude to my thesis supervisor's Ing. Jiri Moravec, Ph.D. and for his continuous support, patience, motivation, enthusiasm, and immense knowledge.

My deepest thanks for my thesis consultant Ing. Jan Skocilas, Ph.D. who has been a major source of motivation, informative and advice. His guidance and knowledge led me to acquire curious interest in plastic production and to complete my thesis in correct direction.

Last but not the least; I would like to thank my family, my parents and specially my friends for not only supporting me in a finical way but also spiritually throughout my life.

Bemmireddy, Santhosh Reddy

2019

## ABSTRACT

This master thesis deals with the literature survey on injection molding machine construction, working and process parameters. And also, optimizing process parameters of the injection molding machine in plastic manufacturing technology to analyze and measure the data experiments and simulations on mold flow in the Injection molding machine. The main aim of this thesis is to perform simulations and compare them with machine settings to understand the principle parameters of the injection molding machine by both theoretically and practically for the study of improving product quality.

This involves comparing the Practical results of the parameters of the injection molding machine (flow time, injection pressure, mold temperature, Melting temperature, etc.) by the results of software simulation. Hence we get to know the parameters to be changed to get better quality of the product.

## Scope of thesis work

- Study and understand the concept of plastic production.
- Analyzing the flow of plastic and its parameters in various stages of the injection molding machine.
- Performing experiments with the Software Autodesk Moldflow to get the recommended process parameters for improving the production and quality of the product.
- Designing the mold cavity with runners and sprue of an injection molding machine.
- Plastic flow time analysis in a mold cavity in different cases.
- Analyzing the real technical parameters of an injection moulding machine.
- Comparing the recommended setting of process parameters obtained by simulation in software with the machine real data.

# Contents

1 Introduction .....	11
2 Types of Injection Molding Machine .....	12
2.1 Hydraulic Injection Moulding Machine.....	12
2.2 Electric Injection Moulding Machines .....	12
2.3 Mechanical Injection Moulding Machines.....	13
2.4 Hybrid Injection Moulding Machines .....	13
3. Construction and workingof an injection moulding machine .....	13
3.1 The Plasticizing and injection unit: .....	14
3.1.1 Types of Injection molding screws.....	15
3.2 The Clamping unit .....	17
3.3 The Mould cavity .....	19
4The main process parameters of injection molding .....	22
4.1 Injection speed.....	22
4.2 Pressure .....	22
4.2.1 Injection Pressure .....	22
4.2.2 Holding Pressure .....	23
4.2.3 Clamping Pressure .....	23
4.2.4 Nozzle Pressure.....	23
4.3 Temperature .....	24
4.3.1 The Melting temperature/Barrel Temperature.....	24
4.3.2 Mold temperature .....	24
4.3.3 Nozzle Temperature .....	25
4.4 Time Parameter .....	25
5 Mold Cycle.....	25
6 Model, Material and Software selection .....	28
6.1. Software Used.....	29
6.1.1 Moldflow .....	29
6.1.2 Solid works .....	29
6.2 Modelling .....	30
6.3. Material Used.....	31



6.3.1 Characteristics.....	31
6.3.2 Applications of Polypropylene .....	32
6.3.3 Advantages .....	33
6.3.4 Disadvantages.....	34
7. Task 1 (Simulation).....	35
7.1 Aim .....	35
7.2 Modelling / Importing.....	35
7.3 Simulation .....	36
7.3.1 Material Selection.....	36
7.3.2 Injection location .....	36
7.3.3 Gate.....	37
7.3.4 Parting plane & Mold Size.....	37
7.3.5 Runner.....	38
7.3.6 Sprue.....	39
7.3.7 Pre check Analysis.....	39
7.4 Results and discussion .....	40
7.4.1 Plastic flow .....	40
7.4.2. FillTime.....	41
7.4.3. Confidence of Fill .....	43
7.4.4. InjectionPressure .....	44
7.4.5. PressureDrop .....	45
7.4.6. Temperature at Flow front .....	46
7.4.7. AverageTemperature.....	47
7.4.8. QualityPrediction .....	48
7.4.9 Time to reach ejection temperature .....	49
8. Task 2 .....	50
Understanding the real processing data of the machine .....	50
8.1Injection moulding Machine.....	50
8.1.1 Dimensions and setup of the Machine.....	51
8.1.2 Technical data of the machine.....	53
8.2.1 Fill time .....	54
8.2.3 Temperatures.....	56

8.2.3.1 Melt temperature .....	56
8.2.3.1 Mould temperature .....	56
9. Conclusion:.....	57
Table of Figures.....	59
References .....	60

# 1 Introduction

“Injection moulding is a production method for manufacturing plastic parts by injecting molten material into a mold cavity[1]. Injection moulding can be made with different type of materials mainly like metals, glasses, elastomers, confections, and usually thermoplastic and thermosetting polymers” [2].

Injection moulding is mainly known as the most significant process utilized to produce plastic products. Right now, more than 1/3rd of all thermoplastic materials are injection molded and greater than half of all-polymer processing equipment is for injection moulding. This process is more suitable for Mass production.

The past century has gone through the rapid extension of polymers and plastics and their incursion into each and every market. Compared to other materials, plastics are now the most widely used materials, surpassing the world’s consumption of steel, aluminum, rubber, copper, and zinc by weight and volume. Plastic materials and products treat the world economy in a state to profit by a turnaround in many fields like packaging, appliance, transportation, housing, automotive, and many other industries.

“Injection moulded parts need to be very precisely designed for better moulding process. Before thinking about the process, the material used for the part, the desired shape and features of the part, the material of the mold, and the properties of the moulding machine must all be analyzed. The versatility of injection molding is aided by the breadth of design ideas and circumstances.”[3].

Material for the part is filled within a heated barrel, mixed, and injected into a mold cavity, where it cools and hardens in the cavity. After a product is designed, generally it will be done by a designer, molds are made by a mold-maker from metals, steel or aluminum is used. Injection molding is largely employed for producing a different quality of parts, of the minutest (tiny) elements to complete body panels of cars.

The main aim of this project is to analyze the processing parameters of the injection molding to meet the requirements for producing a better quality product. Here I am going to simulate the part with given shape and dimensions to get the results of applicable processing parameters to produce a better quality product. Conclusively I will compare the simulation results with the actual machine parameters, if the given values of the machine differ with the simulation results then we need to change the machine input values to get the better part/product.

## 2 Types of Injection Molding Machine

Injection moulding machines are classified essentially by the type of driving systems they use,

- Hydraulic
- Electrical
- Mechanical and
- Hybrid.

### 2.1 Hydraulic Injection Moulding Machine

Historically, Hydraulic presses have been the only possible choice to molders until Nissei Plastic Industrial Co., LTD (Japanese company) launched the first all-electric injection molding machine in 1983[4].

It is used for different purpose and applications. Which is Available in different sizes, hydraulic machines are more suitable for performing repeated tasks with a fair degree of accuracy. The standard value of these machines is significantly less than its alternatives. Maintenance cost of this machine is very low, because of its parts are stronger and cheaper.

### 2.2 Electric Injection Moulding Machines

This type of machines is also known as Electric Machine Terminologies (EMT) with low operation costs and are more suitable for producing high-end injection molded parts. Mostly suitable for mass production. Electric presses are chosen because of more environmentally friendly, quieter, faster, and have a higher accuracy. Of-course, these machines are too expensive of all but the maintenance cost is low.

Since there is no use of oil for this machine, there is no chance of leakage from oil and also no filters to be replaced. Hence the expenses are less and it is safe for manufacturing medical parts. This machine is digitally controlled, so this is suitable for repetitive process and no need of man supervision.

## 2.3 Mechanical Injection Moulding Machines

Mechanical standard machines use the Mechanical toggle system for making up tonnage on the clamp side of the machine. Mechanical clamps are ideally referred to as toggle clamps. Tonnage is needed on every machine so that these clamp side of the machine behaves not to open due to the injection pressure. It will create flash in the plastic product if the tool half starts to open up[4].

## 2.4 Hybrid Injection Moulding Machines

This kind of machines are also more known as "Servo Hydraulic Molding machines". These machines are well recognized for the production in thermoplastics processing. These machines combine the electric power and accuracy of an electric machine with the power and dynamics of hydraulic machines for the best performance. Producers and suppliers of molded machines concentrate on producing the best quality machines that which gives high performance, reliability, safety, and cost-efficiency. Since The product variety is the most important need of the customer which allowed makers to go from hydraulic machines to electric and hybrid mold machines.

Injection molding is commonly used and essential modern manufacturing method. It is a little bit similar to die casting but the exception is in the raw material which is used is different. In die casting, we utilize metals which need very high temperature for melting but injection molding is essentially made on glasses, elastomers and most generally on thermoplastic and thermosetting polymers but this technique is extensively applied for the invention of thermoplastic materials.

## 3. Construction and working of an injection molding machine

Injection molding is an essential modern manufacturing method. It is a little bit similar to die casting but the exception is in the raw material which is used is different. In die casting, we utilize metals with very high temperature for melting but injection molding is essentially made on glasses, elastomers and mostly on thermoplastic and thermosetting polymers but this method is extensively applied for the invention of thermoplastic materials.

## Process and equipment:

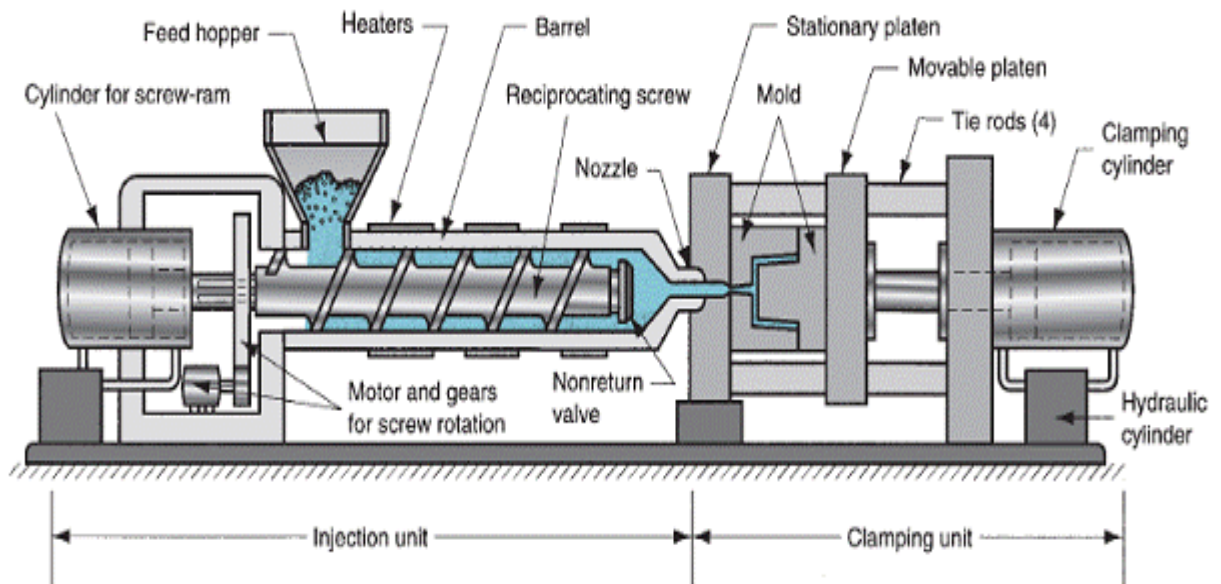


Figure 1 Schematic of a reciprocating screw injection molding machine [5].

The construction of an injection moulding machine can be explained as three main components which are shown in the fig (1).

- The Plasticizing and injection unit.
- The clamping unit.
- Mould cavity.

### 3.1 The Plasticizing and injection unit:

The main responsibility of this system is heating/melting of a polymer to a highly plastic state and injecting it into the mould cavity by pushing it to pass under high pressure at a particular

temperature without any variation in their chemical structure and then solidifies.

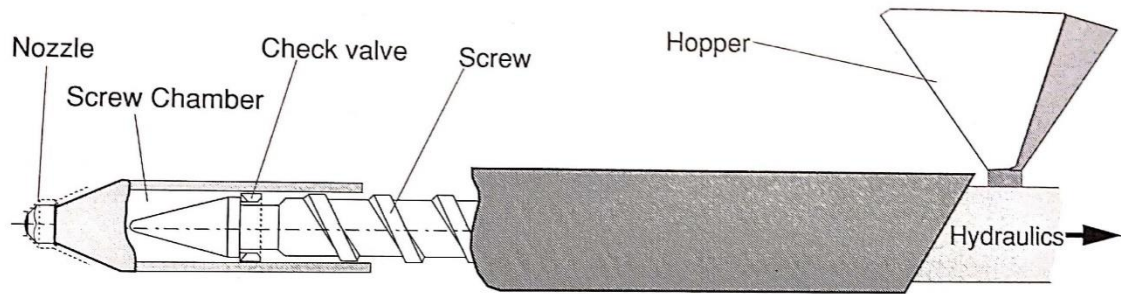


Figure 2 Plasticizing unit [6]

The main elements of the injection unit are,

- Hopper
- Screw
- Heating bands
- Check valve
- Nozzle.

The hopper, heating bands and the screw are identical. In case of single screw extruder except the fact that the screw here in this machine can move back and forward to melt growth and injection. Hence it is called a reciprocating screw. The best stroke in a reciprocating screw is 3 times the screw diameter.

Polymer pellets are supplied through the hopper utilizing gravity, pass through the cooled throat and fall into the rotating screw. When the motor starts and gives motion to the screw, friction will be generated from the rotating screw and heating bands melt the polymer material. At the edge of the screw, molten material moves within a non-return valve which acts as a plunger while the injection and packing. This will stop the melt to flow back into the screw channel and to the hopper. When enough melt is filled in between the nozzle and non-return valve then the screw stops rotating or the motor will get stopped automatically [6].

### 3.1.1 Types of Injection molding screws

There are no rules in the plastics industry for a general-purpose injection moulding screw design. The selection of these injection screw design for a particular process will be generally based on the particular part to be moulded or going to be produced. Where it depends on the properties like part material, weight, and size and wall thickness.

Selection of the proper molding screw is more important for manufacturing better quality products with maximum output.

“Choosing a screw without knowledge of the parts is like buying a car without any preference for performance and handling requirements”[7].

#### 3.1.1.1 General purpose screws

The benefit of a general purpose screws is that they can be used for the largest variety of plastic materials such as PP, PE, Nylon and PC (Polycarbonate). Where these are extremely flexible and suitable for moulding companies that mold a diversity of materials.

The main difficulty is that, for some materials, part quality and productivity rates will be very low compared to the advanced injection molding screw designs such as the barrier or Double flight screw.

#### 3.1.1.2 Double Flight Screws

These type of screws are typically meant to deliver a high quality melt at fast rates.

The design assures the plastic is completely melted before it enters the compression zone, which is different in the case of general purpose screws.

This type of screws can also be utilized in technical parts from PP and PA with thin wall technical parts, which do not plasticize properly with other types of screws.

#### 3.1.1.3 Wear-resistant screws

Applying a heat-treated screw and barrel which will give better life than non-heat-treated parts. Where this is very essential when the material have some level of reinforcement as this is mostly abrasive and may wear out the screw and barrel quickly than the material without reinforcement.

If the screw and barrel start to wear, the part quality will start to suffer and it will only be a matter of time before a replacement will be needed. This is too costly not in the point of replacement of the screw but also it gives loss in the time and production [8].



### 3.2 The Clamping unit

The main function of this unit is to open and close the mold, holds the mold tightly to stop from flash during injection, packing and holding. There are two types of clamping units which are mostly used in the modern injection moulding machine. They are Mechanical and Hydraulic and a combination of both.

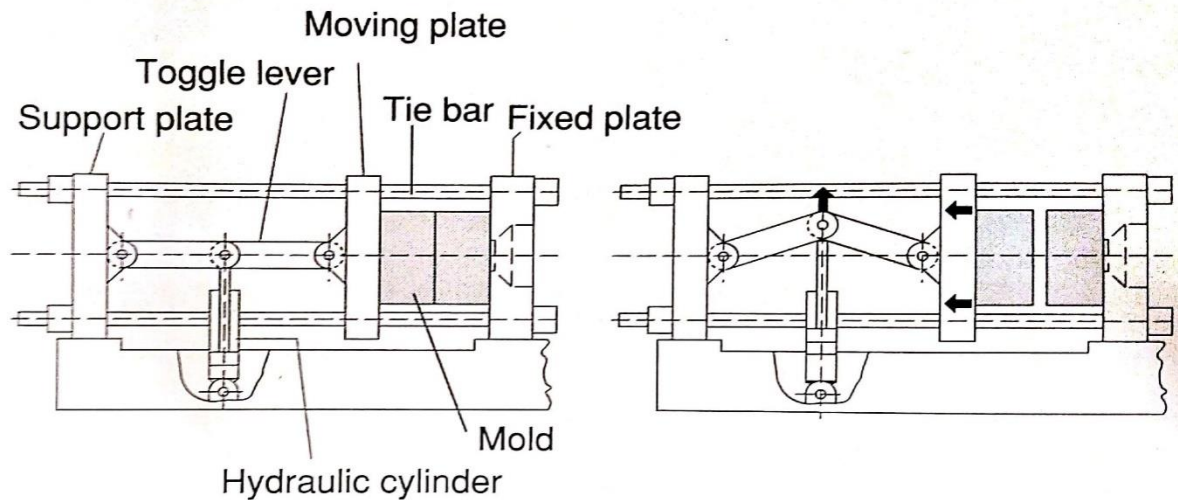


Figure 3 Clamping unit with a toggle mechanism [9].

The toggle is a mechanical device which is actuated by a hydraulic unit which is shown in the fig 4. The main advantage of the toggle mechanism is that as the mold awaits closure, the potential closing force rises and the closing decelerates. Although, this device just gives it's most leading closing force when the arrangement is fully extended. The above fig shows the schematic of a toggle in the open and closed situations.

The main advantages of the hydraulic system over the electric machines are that a maximum clamping force is obtained at any mold closing situation and that the system can use many mold sizes without significant system changes [9].

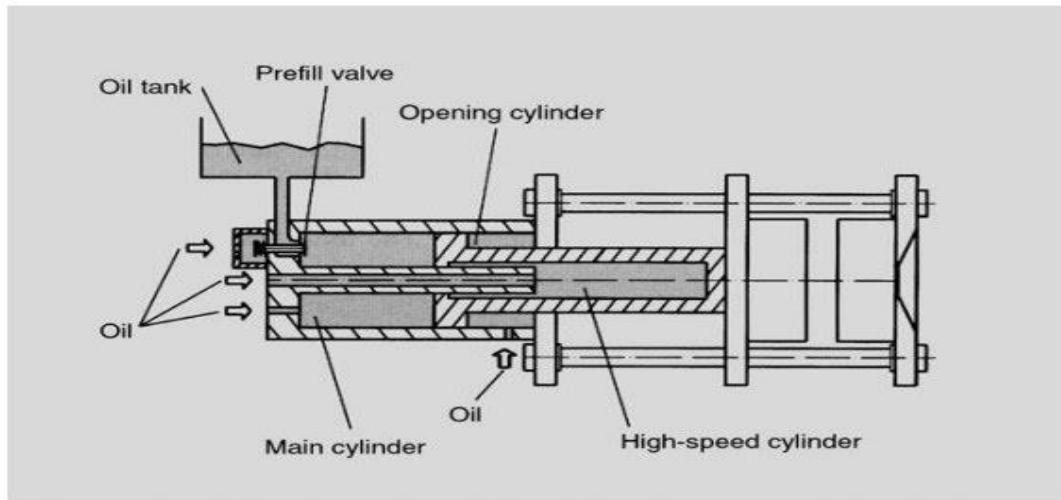


Figure 4 Hydraulic clamping unit [10]

### Comparison of Toggle and Hydraulic clamping unit

The main advantages, disadvantages and limitations of these clamping units is given below in Table 1.

Table 1 Comparison of Toggle and Hydraulic clamping unit [11].

	Toggle	Hydraulics
Mould stress	High	Balanced Stress distribution
Mould flow pressure protection	Poor, Resulted from toggle design.	Excellent programmable.
Machine size Limitation	Only up to 1000Tons	No limits
Maintenance cost	Expensive Replacement of Self lubricated bushings.	Low cost, very good replacement of every 5 years.
Cleanness	Poor due to necessity of toggle lubrication grease	Excellent, Self-Lubricating solutions.
Dry cycle timing	Excellent but only for limited sizes of machines	Very good, improved with direct intake throat.
Straightness and Alignment	Not perfect due to clearance of Arms.	Excellent less than 1%.

### 3.3 The Mould cavity

The principal point in an injection molding machine is the mould cavity. The mould distributes polymer melt into the cavities, shapes the part, cools the melt and ejects the finished product. As shown in the Figure 5. The mould is custom made which can be built by mould makers and consists of following elements,

- Sprue and runner system
- Gate
- Mould cavity
- Cooling system
- Ejector system

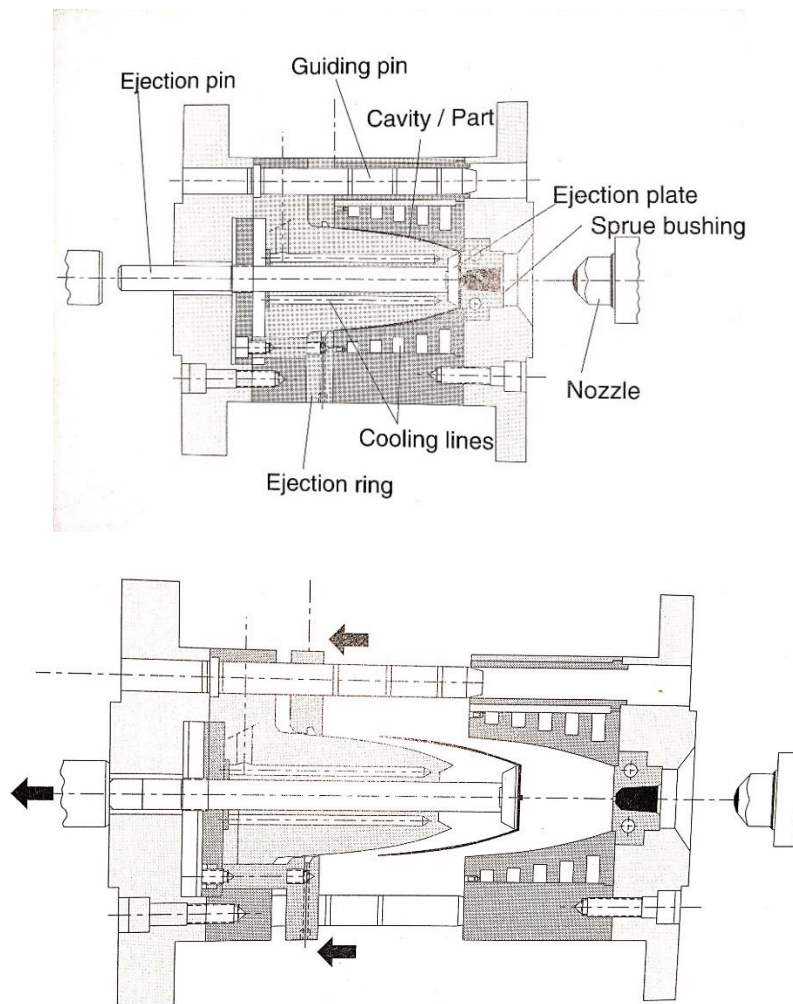


Figure 5 Injection Mold Cavity[12].

When mold filling, the melt flows through the sprue and is distributed into the cavities by the runners, as shown in the Fig 6.

The runner system in Fig.6. (a) is symmetric where all cavities are filled at equal time and get the polymer to fulfill all cavities equivalently. The conditions of this runner system are that the flow paths are long, reaching to large material and pressure loss.

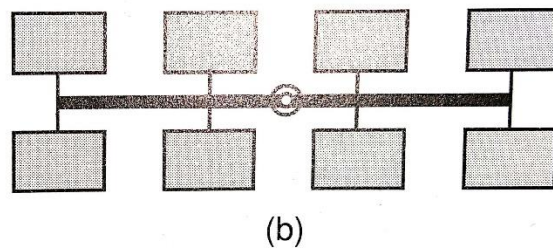
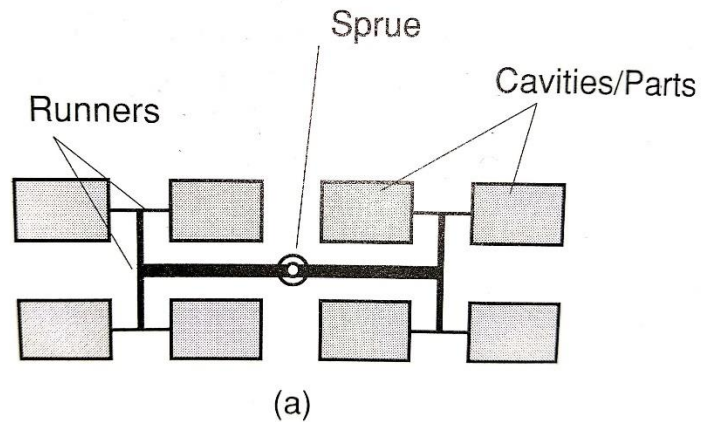


Figure 6 Schematic of different runner system arrangements<sup>[12]</sup>.

On the other hand, the asymmetric runner system is shown in fig.6.(b) Leads to parts of various quality. Equal filling of the mold cavities can also be done by varying runner diameters.

There are two sets of runner systems called hot and cold runners. Cold runners are separated with the part and are outfitted after mold elimination. The advantage is a more moderate mold cost. The hot runner operates the polymer above its melt temperature. The material lingers in the runner's system after ejection and is injected into the cavity in the following cycle. Although a hot runner system is considered to be advances mold cost, its advantages include the removal of trim and more moderate pressures for injection and production of waste.

Commonly, there are several types of gates utilized according to the need and part to be produced. Sometimes sprue labor's as the gate when big parts are injection molded as shown in fig 12. The sprue must be trimmed, usually needing further surface finishing. On the other had pin type gate is a very small opening that combines the sprue and the runners to the mold cavity. Other types of gates are also shown in the figure. Film gates, which are utilized to reduce orientation, and disk or diaphragm gates which are applied for symmetric parts such as compact Discs [12].

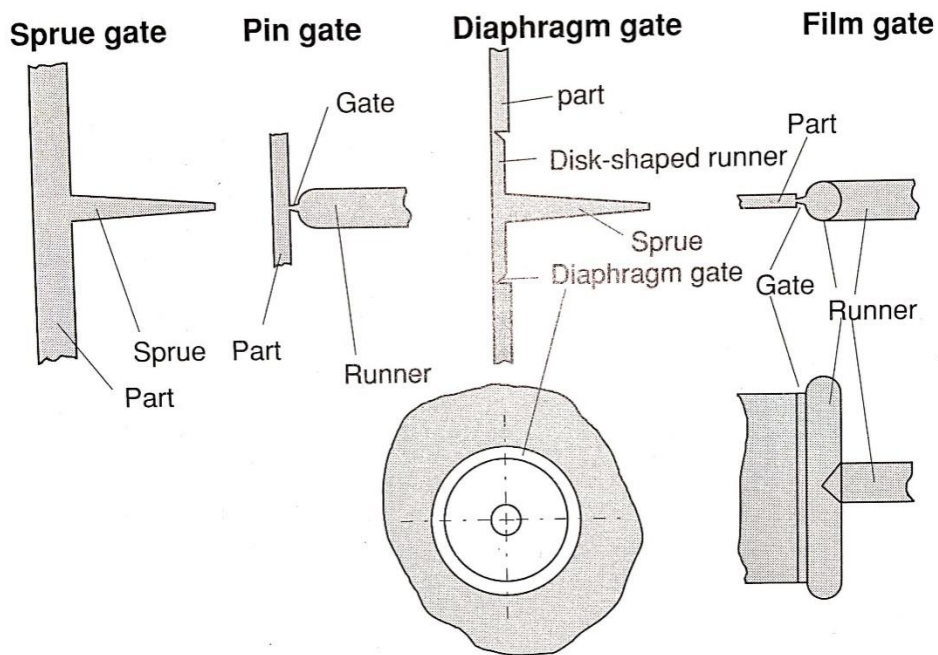


Figure 7 Schematic of different gating systems [12].

## 4 The main process parameters of injection molding

Injection molding process of plasticizing flow and cooling conditions depends on temperature, pressure and the corresponding time which implies the quality of plastic parts. While adjusting the process parameters, in principle according to the order of the pressure-time-temperature should not change all at the same time. Two or more parameters, to check the flocculent process conditions because plastic parts quality will not be the same. Main process parameters of injection are as follows,

- Injection Speed
- Pressure
- Temperature
- Time parameter

### 4.1 Injection speed

Melt filling, mold time and flow pattern is the most influential in the process of flow conditions. The setting of injection speed has complete control of product image quality (it has to be accurate). Injection speed setting is the fundamental principle of meeting plastic flow inside the cavity, according to its movement formed by the cross-section size to lift and remain by the slow to fast. Where low pressure will reduce the internal stress of plastics and it will develop the strength. Select high low-speed feeding can cause the flow smoother, little shear rate, plastics stability, prevent shrinkage error [13].

### 4.2 Pressure

The pressure is found mostly in the injection area, but there is also pressure found in the clamp unit of the molding machine. We will discuss all of these pressure requirements here.

- Injection pressure
- Holding Pressure
- Clamp pressure
- Nozzle pressure

#### 4.2.1 Injection Pressure

Different injection pressure needed for plastic products, such as PA, PP material, where as rising pressure will significantly increase its liquidity, injection pressure defines the density of

the product, namely the gloss appearance, Melt forward to overcome the resistance, a direct impact on product size, weight, and deformation, etc., it has no fixed/standard value, and the more difficult the mold filling, the injection pressure will be increased. Flow nature of each type of plastic is different, also different for the same material with change in melt temperature, different products, mold design, mold temperature will change the resistance of the material flow. In a variety of different conditions to maintain the same injection speed, injection pressure would surely have to change, make it overcome the resistance of the melt flow caused by Injection pressure. The higher the pressure, the higher the stress, and the greater the reaction when it is released. So, we should determine the minimum amount of pressure necessary to fill the mold, and then use all of it. And, the hotter the plastic, the more fluid it becomes and the lower the pressure can be to fill the mold [14].

#### 4.2.2 Holding Pressure

Once the majority of the plastic (95%) has been injected using regular injection pressure, the machine should drop into hold pressure. This pressure is about 0.5 times the injection pressure and is utilized to finish filling the mold by packing the molecules together in an orderly fashion. Hold pressure is required until the gate freezes off, normally it will take 3 to 4 seconds. Once that passes, hold pressure has no more effect on the molecules on the other side of the gate. If the hold pressure is delivered before the Gate freezes, the material in the cavity is still molten and will be absorbed back out of the cavity. At the very least, there will be inadequate pressure to pack the molecules commonly and odd shrinkage and cooling will take place.<sup>[14]</sup>

#### 4.2.3 Clamping Pressure

At the other end of the machine, we have clamp pressure. The only purpose to have clamp pressure is to fix the mold closed tight against injection pressure. Therefore, the amount of clamp pressure required is based on the material to be molded. The easier flow materials require less injection pressure, thus they require less clamp pressure. Similarly, the stiffer flow materials will require more injection pressure, so more clamp pressure.

#### 4.2.4 Nozzle Pressure

Nozzle pressure indicates the pressure contained within the nozzle. It is the pressure which makes the material to flow. This pressure does not have a fixed value but increases based on the increasing complexity of mold filling.

There is a direct connection between injection pressure and nozzle pressure. In screw machines, the nozzle pressure is about 10% less than the injection pressure. Pressure losses in ram machines are much higher [15].

## 4.3 Temperature

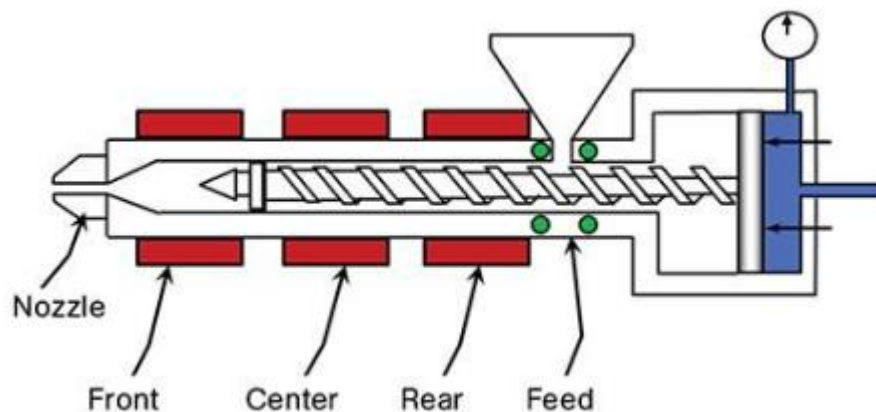
Temperature also plays the vital role in Injection molding process. There will be three types of temperatures required in the process they are given as,

- Melting temperature/Barrel Temperature
- Mold temperature
- Nozzle Temperatures

### 4.3.1 The Melting temperature/Barrel Temperature

The melt temperature plays the key role for the flow behavior of the mould, the melting point is the temperature of the molten state. The plastic with different molecular structure and composition have different influence on its liquidity, depends on the temperature.

A major section that gives the importance of temperature control in injection molding is the barrel of the machine. The plastic travels through usually have between 3 and 5 individual heating zones called barrels which is shown in the Fig 8.



*Figure 8 Barrel Zones [16].*

The material we are using requires the barrel temperature. The temperature settings are defined by the material's supplier. Therefore, the machine's temperature settings should change with the material being used for the production [16]. For example, it should be 210° C to 230° C for polypropylene.

### 4.3.2 Mold temperature

The significance of temperature control in injection moulding is the mould itself. Managing optimal mold temperature decreases unit costs, gives the quality of the product, and improves uniform moulding of parts [17].



Mold temperature can influence quality of the in different ways. Too low temperature can produce knit lines and can give an incomplete part. Too high of a temperature can create warping or blistering in the part.

The mold temperature should be less than the temperature of the barrel. This will allow the material to cool down. The temperature in the mold is generally between 65 and 130° C<sup>[17]</sup>.

Maintaining optimum mold temperature for the material used will improve the part quality, lower production cost, improves the accuracy of parts, lower part distortion, and reduce the time it takes to cool down [17].

### 4.3.3 Nozzle Temperature

The nozzle area of the injection mould is also one that shows the importance of temperature control in injection molding. Make sure that the nozzle temperature should be less than the mold temperature. If the nozzle temperature is more than the mold it may cause the plastic to drool. If the temperature in the nozzle is too low, then it can create the plastic to decompose and possibly block the nozzle [18].

## 4.4 Time Parameter

Injection time and cooling time has a decisive influence on the quality of injection molded parts. Mold filling time is generally not more than 10 s (mold from sprue to the whole cavity). The whole process will take around 1 min where it depends on the holding time. The holding time is longer, the plastic pieces of wall thickness take a long time, to ensure minimum contraction. The cooling time depends on plastic crystalline factors like material thickness, mold temperature adjusted according to the specific situation [19]. Molding cycle is as follows shown in Fig 9.

## 5 Mold Cycle

The sequence of events during the injection molding of a plastic part, as shown in Fig9.Is called the injection molding cycle.

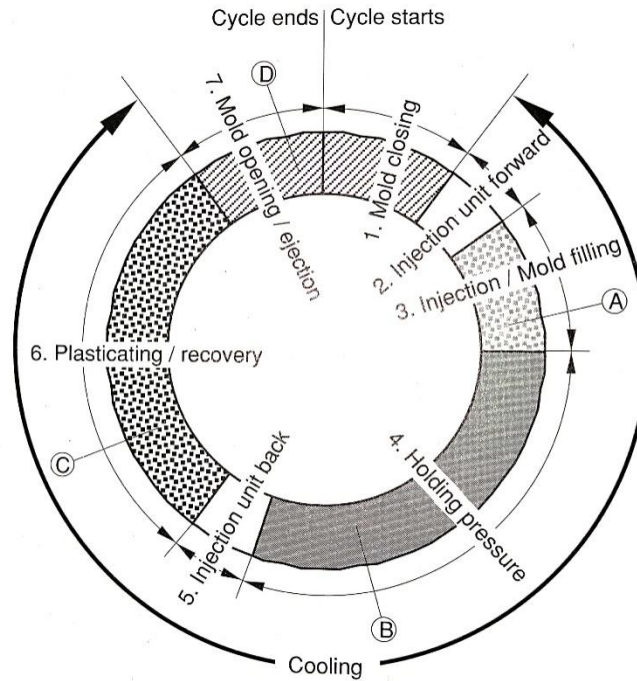


Figure 9 Injection Molding cycle [20].

Typical process cycle time varies depending on the part weight, part thickness, material properties, machine setting specified to a given process.

The injection molding cycle can be separated into different stages.

They are mold closing, filling, packing, cooling, and molding opening and injection as shown in the Fig 10.

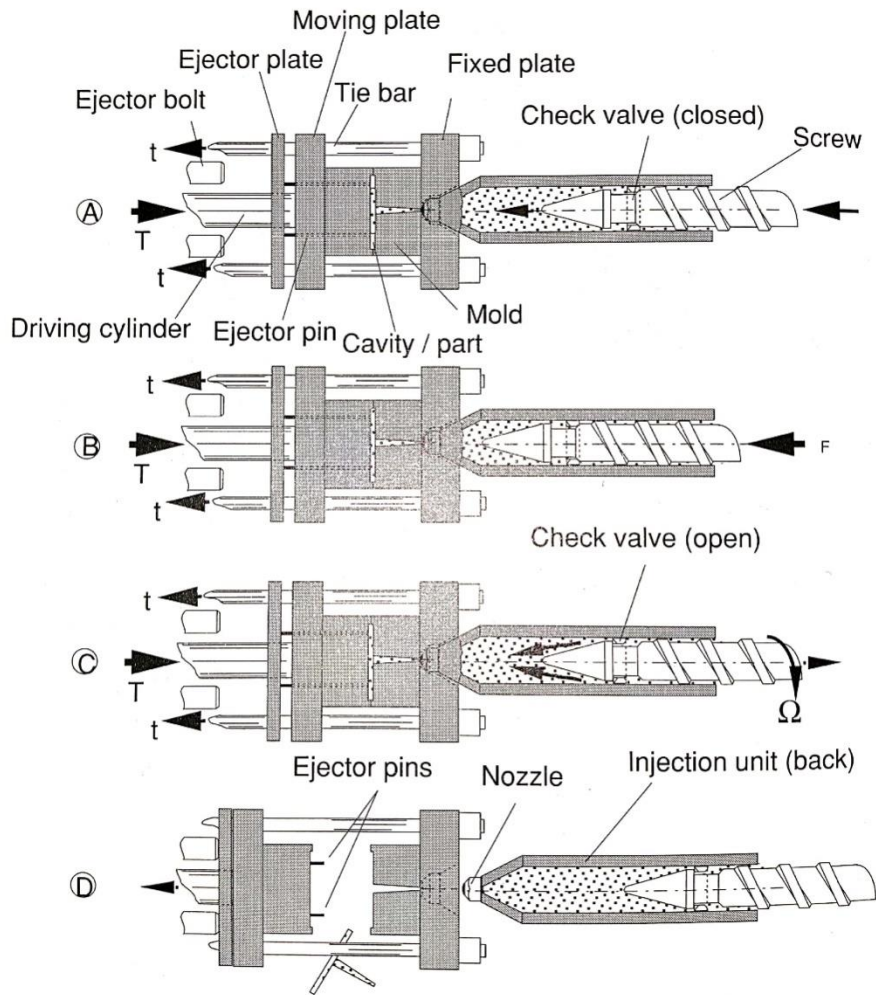


Figure 10 Sequence of events during an injection moulding cycle[20].

The cycle starts when the mold closes, followed by flow of the polymer into the mould cavity. Once the mould cavity is filled, a holding pressure is stated to requite for material shrinkage. In the next step, the screw rotates by means of motor, feeding the next shot to the front of the screw. This makes the screw to retreat as the next shot is ready. Once the part is sufficiently cool, the mold opens and the part is extracted. Figure 10 shows the sequence of performances during the injection moulding cycle. The figure explains that the cycle time is administered by the cooling of the part inside the mold cavity.

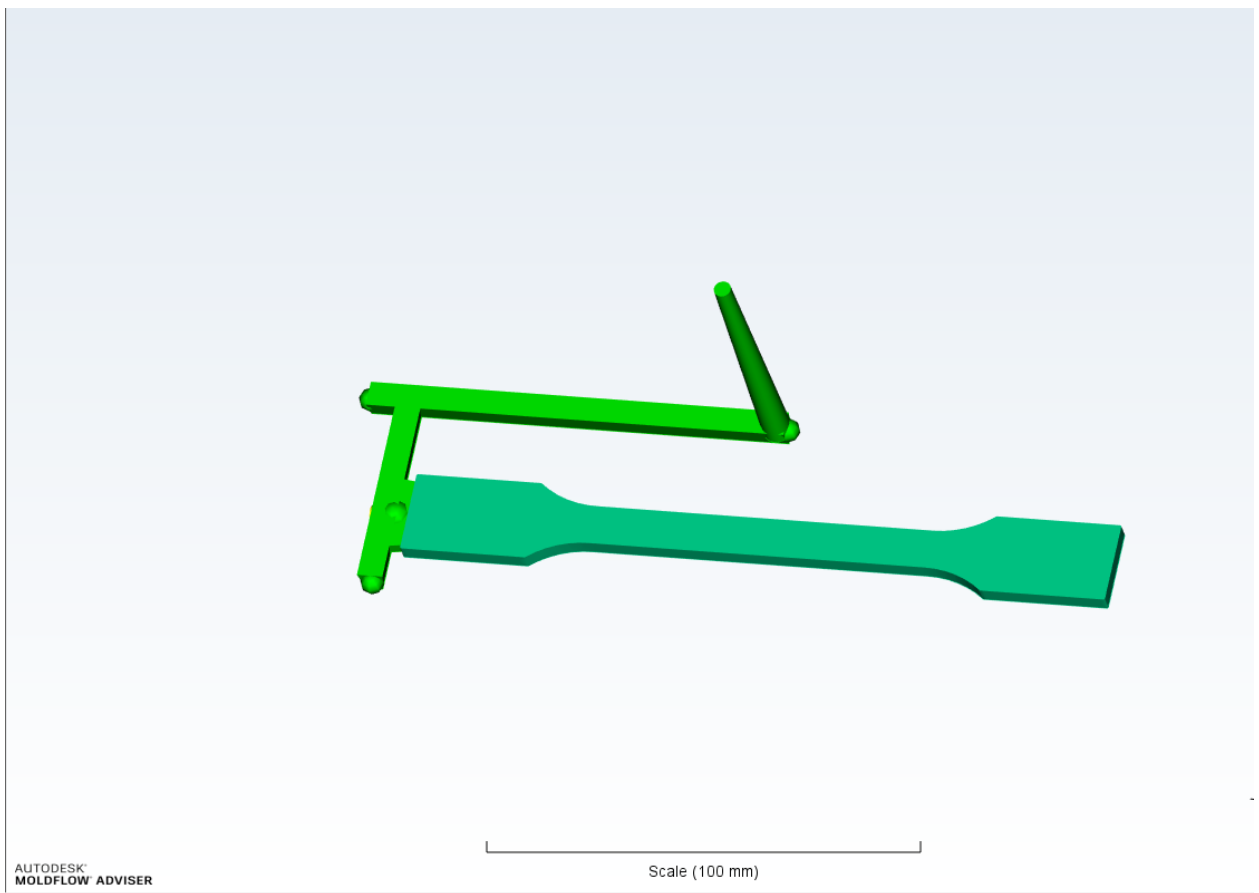
The total cycle time can be calculated, by using

$$T_{\text{cycle}} = T_{\text{closing}} + T_{\text{ejection}} + T_{\text{cooling}} \quad \text{Equation 1[20].}$$

## 6 Model, Material and Software selection

In order to meet the requirements to produce a good quality product by the injection molding process, the most important process parameters are Temperature (Mold, Melt and nozzle), Pressure (injection, Plasticization) and corresponding to the various time ( Injection time, cooling time). Here we are going to study all the parameters for a given model shown in the below fig...By using a software called *AUTODESK MOLD FLOW ADVISER*.

The reason for selecting this part for simulation is, this is the sample part produced by the machine which we have in our university. So I chose this part to analyze and calculate the processing parameters of the Mold in the cavity.



*Figure 11 Sample part.*

## 6.1. Software Used

Solid works, AUTODESK MOLD FLOW.

### 6.1.1 Moldflow

Mold flow, simulation software owned by *Autodesk, Inc.* that produces high-end plastic injection molding engineering software. Moldflow was founded in Melbourne, Australia as Moldflow Pty. Ltd. in 1978 by Colin Austin. In 2008 Moldflow was acquired by Autodesk for \$297M.

Moldflow Adviser presenting manufacturability management and directional feedback for standard part and mold design and Moldflow Insight which presents definitive results for flow, cooling, and warpage along with support for concentrated molding processes. Also, Autodesk produces Moldflow Design, Moldflow CAD Doctor, Moldflow Magic's STL Expert, and Moldflow Structural Alliance that serve as connectivity tools for other CAD and CAE software. They also own a free results spectator, Moldflow Communicator <sup>[21]</sup>.

We also have software's like solid works plastics which is made particularly for plastics. The reason for me to do this task in Autodesk mold flow adviser is, they are providing the student trail version for free with license whereas it's not free for students in solid works.

### 6.1.2 Solid works

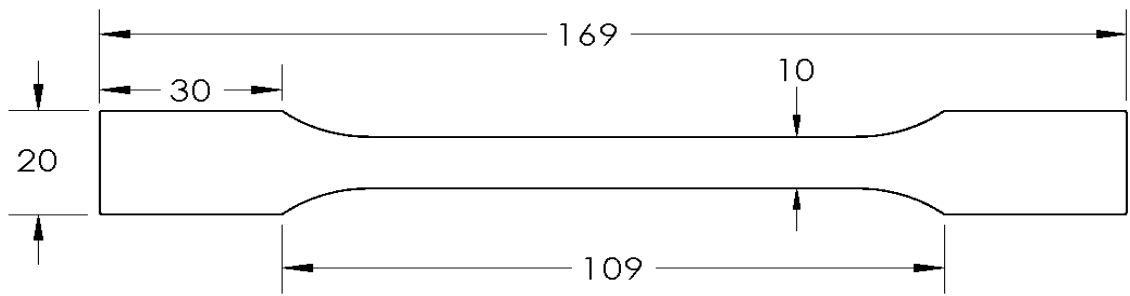
I have used solid works for designing the Part Mold cavity in 3D as the first step before going for simulation in Mold flow Adviser.

## 6.2 Modelling

Software used: Solid works.

The actual geometry of the product was designed in SolidWorks without Gate, runners and sprue and then imported into Autodesk Mold Flow Adviser. In Moldflow we can design the runners and sprue according to the requirements and shapes. This makes the process very simple. The selection of the material is very important for any process here we will be having almost all type of plastic materials already registered in the software hence we can select the material accordingly.

The Part design with dimensions are shown in the following fig...



12 Figure Sample part front view

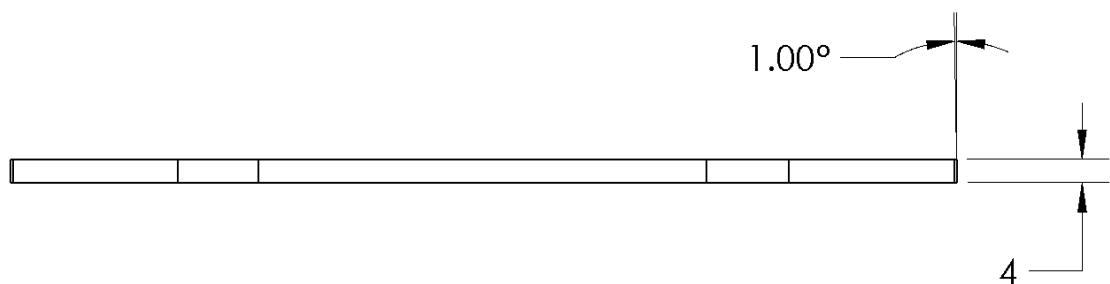


Figure 13 Sample part Top view

**Note:** All the dimensions are in mm.

## 6.3. Material Used

### Polypropylene

Polypropylene (PP), also acknowledged as Polypropene, It is delivered via chain-growth polymerization from the monomer propylene. Polypropylene is a thermoplastic polymer utilized in an extended type of applications like the packaging for various industries including the automotive industry, and textiles. After Polyethylene, Polypropylene is the most generally fabricated plastic in the world right now [22].

The important interests of Polypropylene are that it can be converted into a living hinge (a flexible segment of material, normally made from some type of plastic that joins two rigid surfaces). Living hinges are extremely thin pieces of plastic that bend without cracking (even over extreme ranges of motion 360°). They are not especially useful for structural purposes but are particularly useful for non-load-bearing purposes such as the top of a bottle of shampoo.

Another benefit of Polypropylene is that it can be easily copolymerized with different polymers like polyethylene. Copolymerization changes the material properties [23].

Polypropylene PPH Polypropylene PPH 7060 is homopolymer with a Melt Flow Index of 12 g/10 min. Polypropylene PPH 7060 is intended for the injection molding of packaging containers, toys, domestic appliances, garden furniture, and caps & closures.

### 6.3.1 Characteristics

Polypropylene is classified as a “thermoplastic” material which has to do with the way the plastic responds to heat. Polypropylene PPH Polypropylene PPH 7060 is homo polymer with a Melt Flow Index of 12 g/10 min. Polypropylene PPH 7060 is intended for the injection molding of packaging containers, toys, domestic appliances, garden furniture and caps & closures [24].

Some of the most significant properties of polypropylene are,

	Method	Unit	Typical Value
<b>Rheological properties</b>			
Melt Flow Index 230°C/2.16 kg	ISO 1133	g/10 min	12
<b>Mechanical properties</b>			
Tensile Strength at Yield	ISO 527-2	MPa	32
Elongation at Yield	ISO 527-2	%	10
Tensile modulus	ISO 527-2	MPa	1550
Flexural modulus	ISO 178	MPa	1450
Izod Impact Strength (notched) at 23°C	ISO 180	kJ/m <sup>2</sup>	3.5
Charpy Impact Strength (notched) at 23°C	ISO 179	kJ/m <sup>2</sup>	4.5
Hardness Rockwell - R-scale	ISO 2039-2		95
<b>Thermal properties</b>			
Melting Point	ISO 3146	°C	165
Vicat Softening Point	ISO 306	°C	
50N-50°C per hour			87
10N-50°C per hour			152
Heat Deflection Temperature	ISO 752	°C	
1.80 MPa - 120°C per hour			55
0.45 MPa - 120°C per hour			100
<b>Other physical properties</b>			
Density	ISO 1183	g/cm <sup>3</sup>	0.905
Bulk Density	ISO 1183	g/cm <sup>3</sup>	0.525

*Table 2 properties of polypropylene*

### 6.3.2 Applications of Polypropylene

Polypropylene is generally used in different applications due to its good chemical resistance and weldability. Some common uses of polypropylene include [24],

#### 6.3.2.1 Packaging Applications

Good barrier properties, high strength, good surface finish, and low cost make Polypropylene ideal for several packaging applications.



#### 6.3.2.2 Consumer Goods

PP is used in various household products and consumer goods applications including translucent parts, housewares, furniture, appliances, luggage, toys, etc.

#### 6.3.2.3 Automotive Applications

Since it is low cost, superior mechanical properties, and moldability, pp is widely employed in automotive parts. Central purposes include battery cases and trays, bumpers, fender liners, interior trim, instrumental panels, and door trims.

#### 6.3.2.4 Fibers and Fabrics

A high volume of PP used in the market section known as fibers and fabrics. PP fiber is used in a host of utilization including raffia/slit-film, tape, strapping, bulk continuous filament, staple fibers, spun bond and continuous filament. PP rope and twine are extremely strong and moisture resistant very fit for marine purposes.

#### 6.3.2.5 Medical Applications

PP is appropriated in several medical applications due to high chemical and bacterial protection. Also, the medical grade PP displays good resistance to steam sterilization.

#### 6.3.2.6 Industrial Applications

Polypropylene sheets are extensively used in the industrial sector to create acid and chemical tanks, sheets, pipes, Returnable Transport Packaging (RTP), etc. because of its features like high tensile strength, resistance to high temperatures and corrosion resistance.

### 6.3.3 Advantages

1. Polypropylene is willingly convenient and analogously inexpensive
2. It has high flexural strength due to its semi-crystalline nature.
3. Polypropylene has a slightly slippery surface.
4. Polypropylene has an elevated impact on strength.
5. Polypropylene is deeply resistant to consuming moisture.
6. Polypropylene has great chemical stability over a wide range of bases and acids.
7. Polypropylene is a reliable electrical nonconductor.
8. Polypropylene possesses good fatigue resistance [24].

#### 6.3.4 Disadvantages

1. Polypropylene holds a great thermal expansion coefficient.
2. Polypropylene is sensitive to UV degradation.
3. Polypropylene holds lesser resistance to chlorinated solvents and aromatics.
4. Polypropylene is identified to be difficult to paint as it has poor bonding properties.
5. Polypropylene is extremely flammable.
6. Polypropylene is receptive to oxidation [24].

## 7. Task 1 (Simulation)

### 7.1 Aim

The main objective of the task is to calculate the processing parameters of the injection molding (Pressure, Temperature, Time, and Quality) for the given part by using the software (Autodesk Mold flow adviser). Here I did not specify any type of machine I considering as an injection molding machine with single cavity. I have already selected the material so we know the properties of material we need for simulation.

### 7.2 Modelling / Importing

The actual geometry 3d model is designed in solid works according to the dimensions of the mold cavity of the machine that we have in our university. The file is then imported into Autodesk Moldflow adviser for simulation as shown in the following fig.

I have not designed the Runners and sprue here in this modelling. We can design it in the Mold flow advisersoftware according to the requirements.

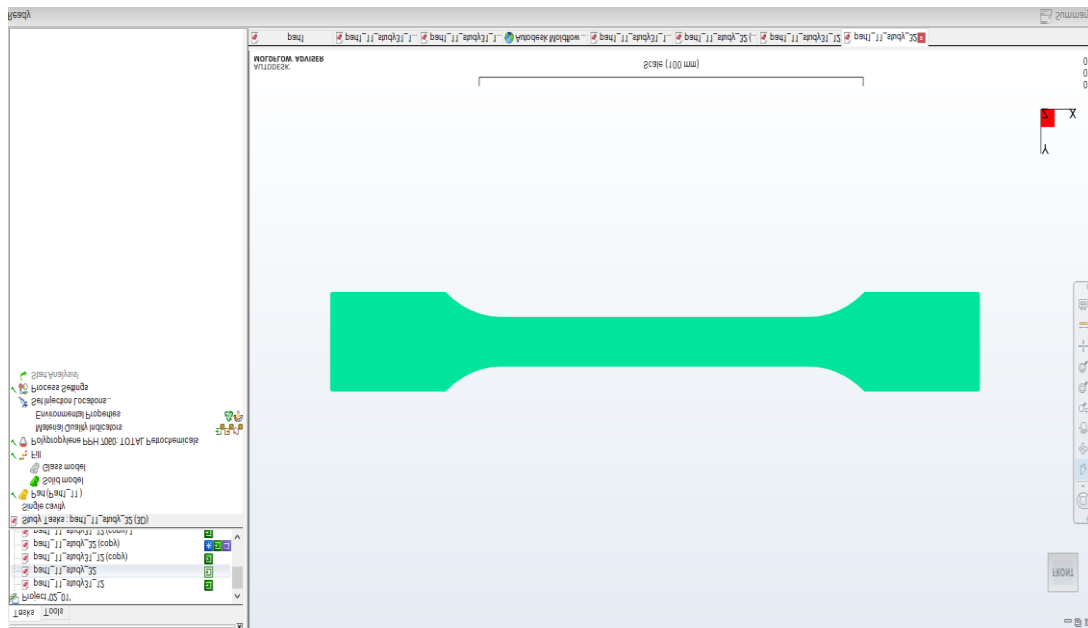


Figure 14 Model

## 7.3 Simulation

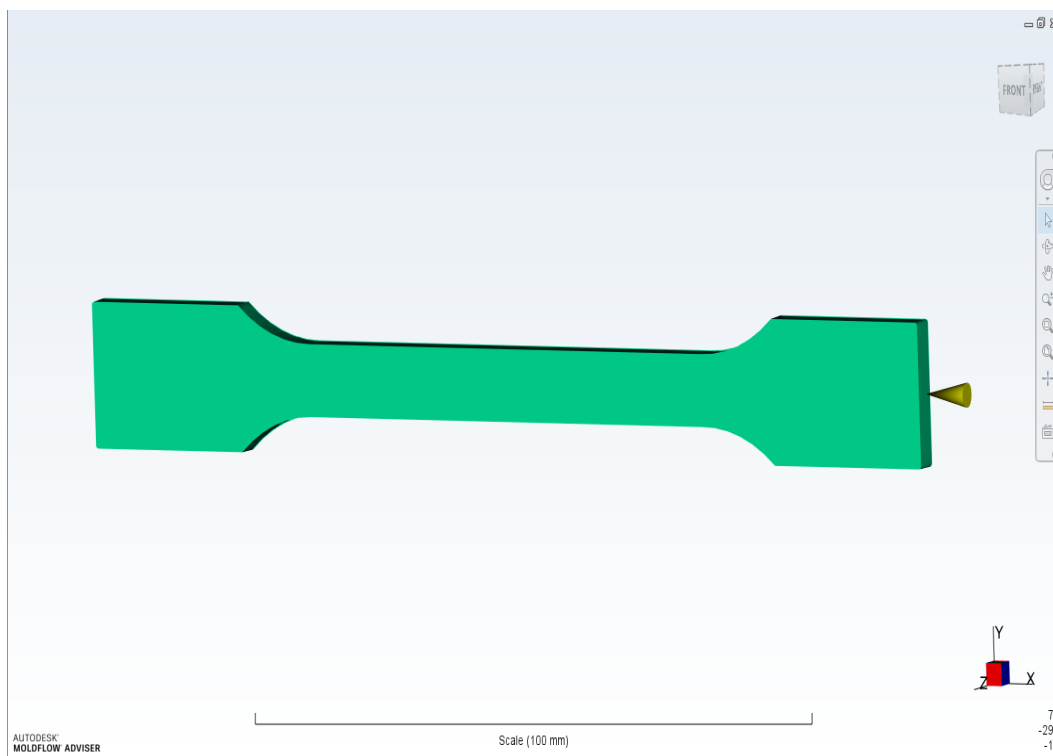
The simulation consists of many sequential steps which are explained below one by one,

### 7.3.1 Material Selection

The first step is selecting a desired plastic material from the software library. Almost all kind of plastic materials are available in the material library of software. I have selected Polypropylene (PPH 7060). It's possible to change the properties of the material by clicking material details.

### 7.3.2 Injection location

The important step after selecting material is setting up injection location. We need to give the location point for injection (gate) for fluid flow into the cavity. This indicates the location of the gate. Since, I know the gate location of the part, I have selected the injection location at the right side of the part by coordinates which is shown in the Fig 13.



*Figure 15 Injection location setup*

### 7.3.3 Gate

We have already given the gate location in the previous step, here we need to provide the shape and dimensions of the gate. My part has a Rectangular tapered gate so I have selected it in the software and since I already know the dimensions of the gate, I have given it here in geometry as show in the Fig 14.

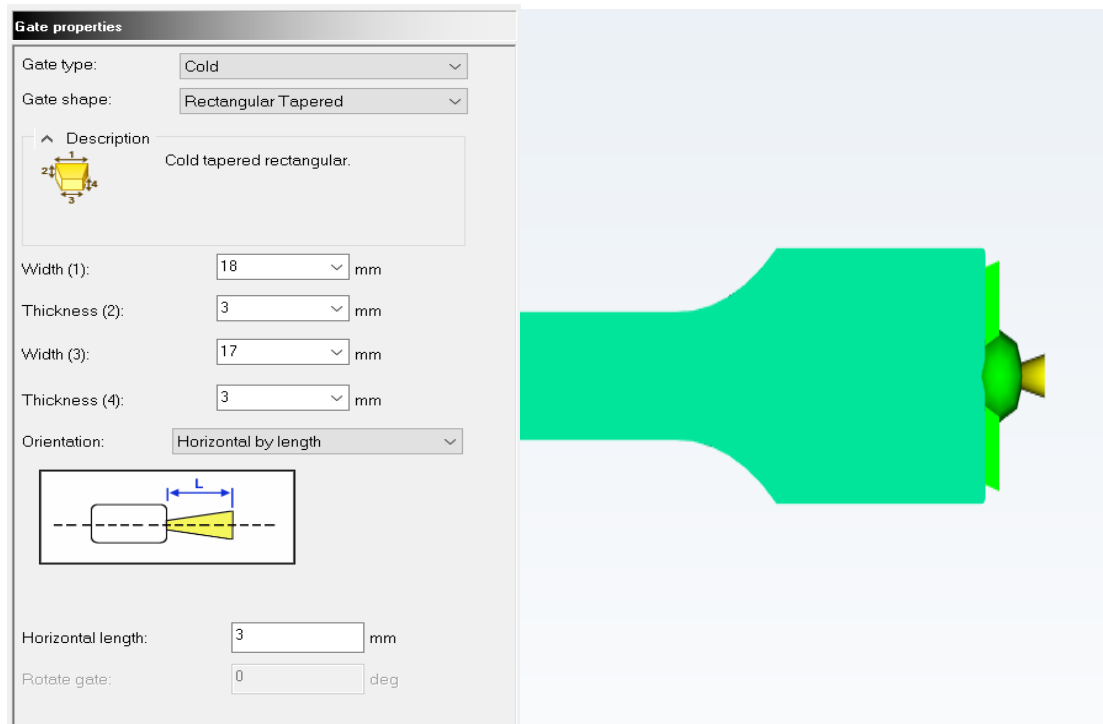


Figure 16 Gate Setup

### 7.3.4 Parting plane & Mold Size

The first is to select the type of the mold we have, either single cavity or multi cavity. In my case I have single cavity. Set parting plane to the part by specifying the depth (Z) we wish to create we can change the plan by dragging it by mouse, the depth value will automatically update. In my case I have the value of z is 1.5mm, I am using this for partition of the part according to the gate mid axis. This partition plane commands the runner's sprue and gate.

The next step is giving mold size, we can say this as meshing. We need to select the mold size option and then it will get automatically generated with some dimensions according to the part. We need to change the size of the mold for giving runners and sprue so I have just extended the mold size in the right hand side where I need to give runners and sprue as shown in the following Fig 15.

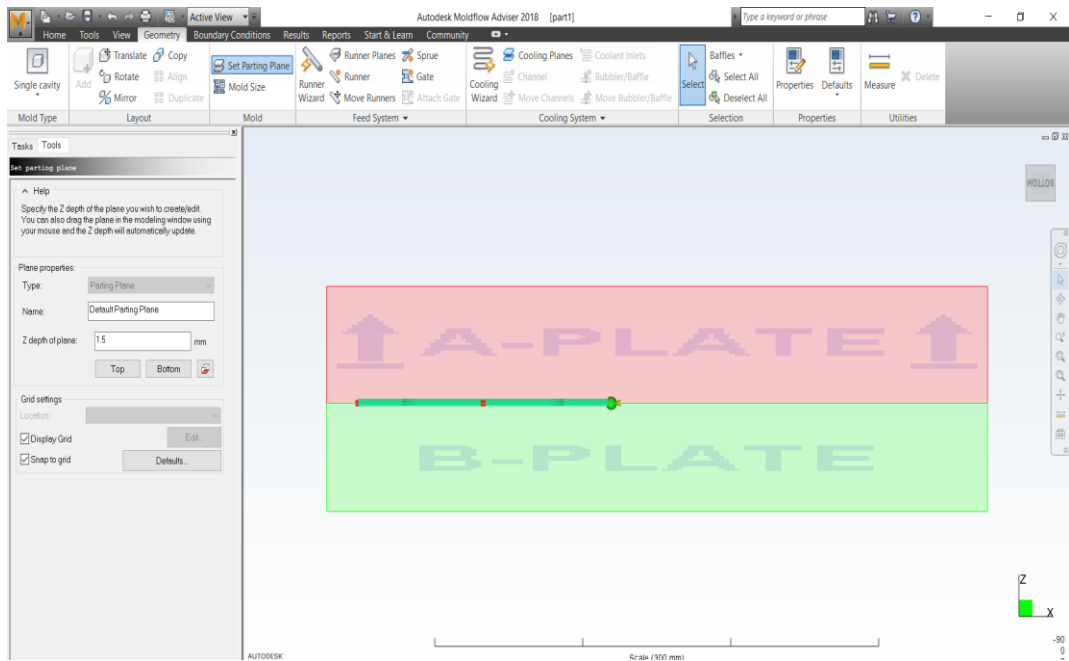


Figure 17 Mold Size

### 7.3.5 Runner

In my case I have two runners which are automatically connected to the gate. These two runners are connected to each other with 90° as given in the following fig...we can select the type of runners (Cold/Hot), shape and dimensions in the software. As I know that I have runners in Trapezoidal shape I have chosen it from the options. The dimensions of both the runners are same except the length. Where the dimensions of the runners are given below. Before giving the runners we should make sure that we have chosen automatically connect runners, it will make sure that the runners are connected to each other without any error.

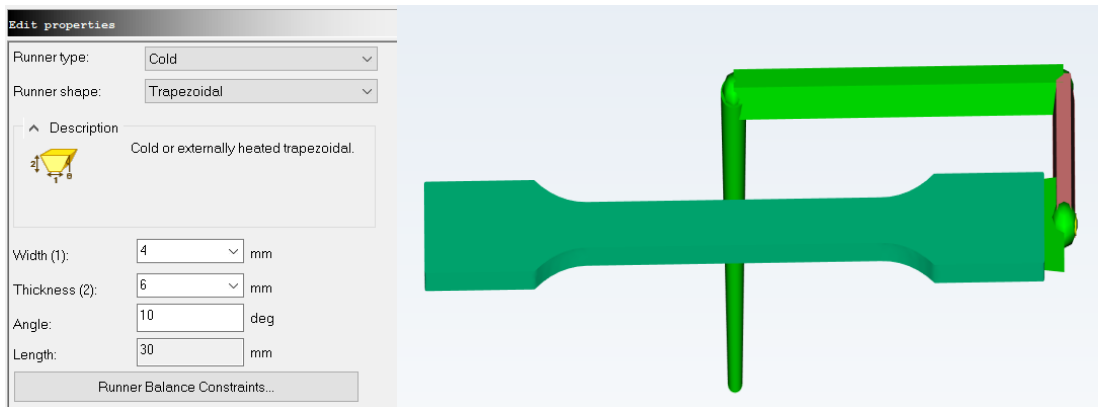


Figure 18 Runner 1

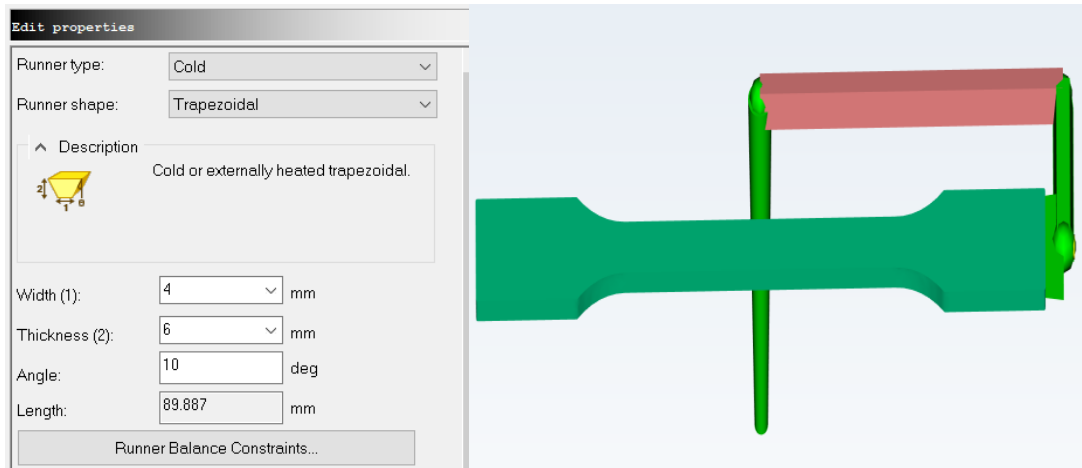


Figure 19 Runner 2

### 7.3.6 Sprue

Sprue is the part or nozzle which makes fluid flow through it into the runners. We need to give sprue before starting analysis. Without sprue we can't run the analysis wizard. We need to select the location on the runner where we need sprue. The part has a sprue in circular tapered shape. So I have chosen it from the options and specified the dimensions as shown in the Fig 18.

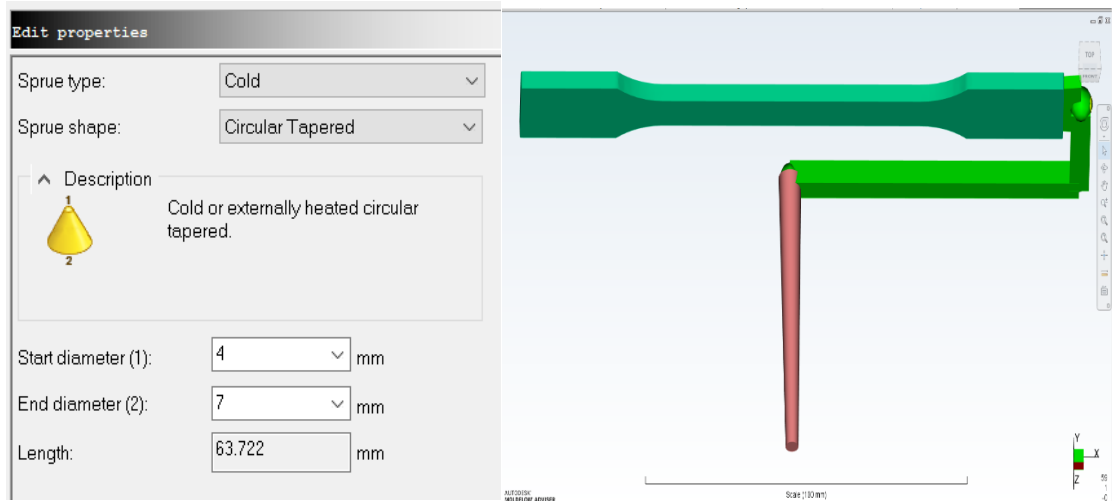


Figure 20 Sprue

### 7.3.7 Pre check Analysis

This is the final step of the simulation. We need to check the analysis before starting the analysis/results wizard. Pre check analysis will show the mistakes and errors we had given. Then we can start the results wizard for the complete analysis to calculate the Parameters.

## 7.4 Results and discussion

### 7.4.1 Plastic flow

The plastic flow results give the flowing nature of material inside the mould cavity. The flow depends on the shape of the mould. The following Fig. 19 shows the filling nature of the plastic material in the mould cavity. The actual result we get here is an animation video of plastic flow. Since, I cannot upload a video where I took a screenshot of the process and imported as a picture. However, I have uploaded the animation video of plastic flow in [YouTube](#) for the plastic flow in the sample product being produced.

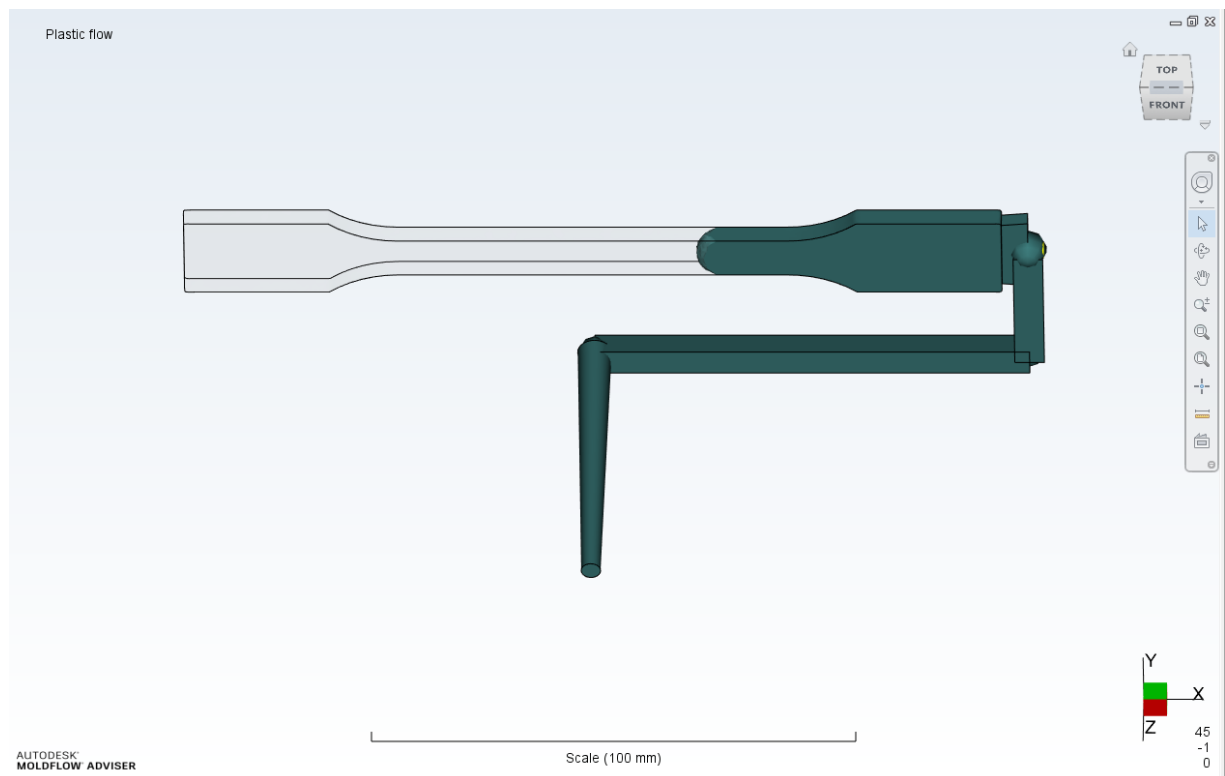


Figure 21 Plastic flow



## 7.4.2. FillTime

The Fill time result provides the status of the flow front at proper periods as the cavity fills. The following fig presents the contour colors that represent the flow of plastic into the part. Total fields with the same color are filled concurrently. The result is dark blue at the injection (sprue) and its red in the last to fill areas. There will be no color if the part isn't filled.

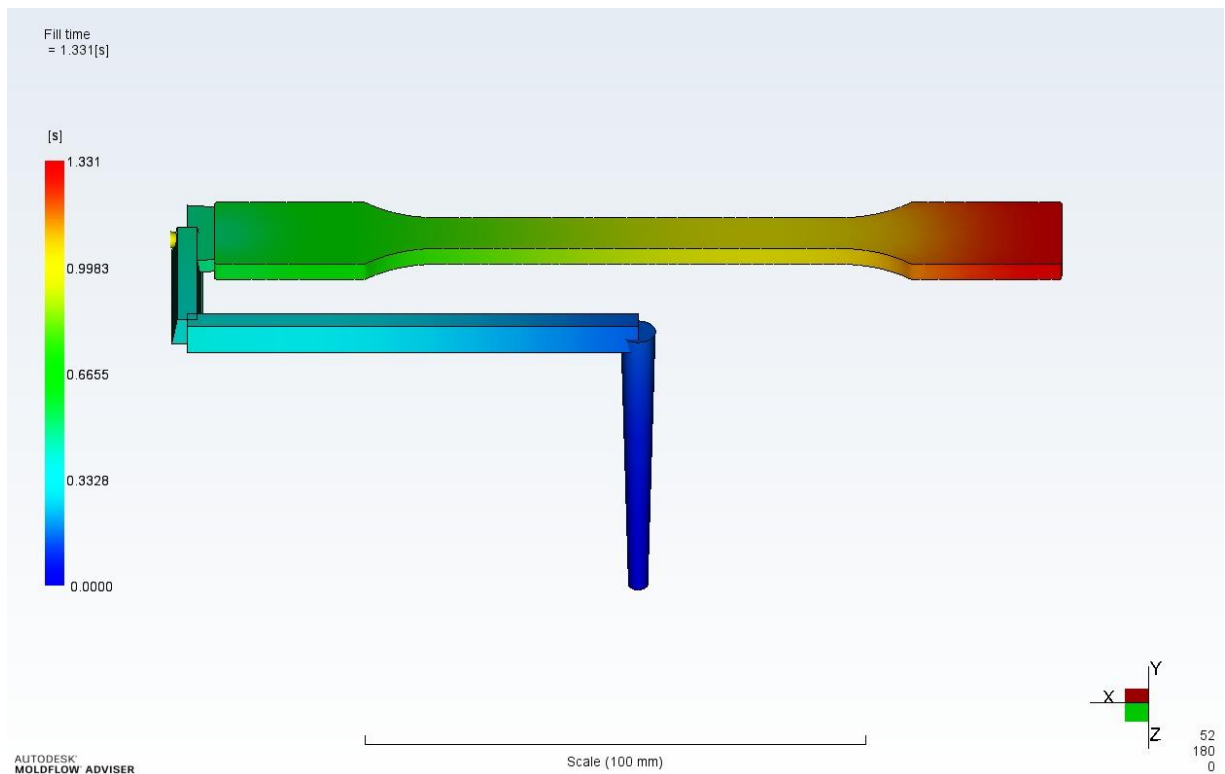


Figure 22 Fill time.

The flow pattern is arranged in the part with a good fill-time result. All paths end at the same time and reach the edges of the model concurrently. Every flow path should finish with red contours. The contours are evenly spaced and show the speed at which the polymer is resulting in broadly arranged contours show rapid flow (Green in the above fig). Narrow contours indicate the part is filling slowly (Yellow) <sup>[25]</sup>.

The filling time is calculated automatically according to the part and material properties, where it takes 1.331s for the mould fill all over the body. As shown in Fig 20.

The following figures shows the flow of material at particular time and area. As we can see it takes around 0.5 seconds for the mould to reach the part from sprue and runners.

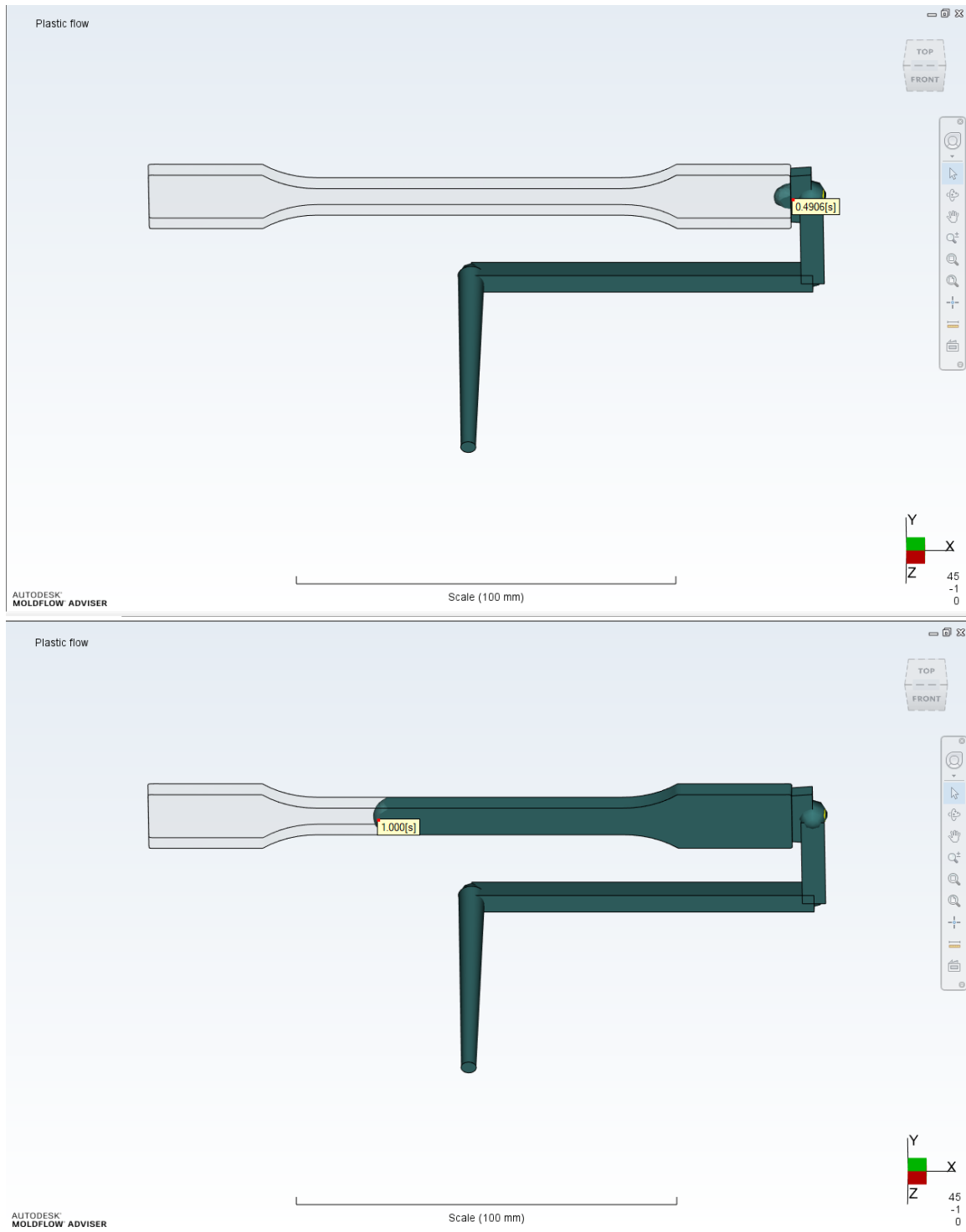
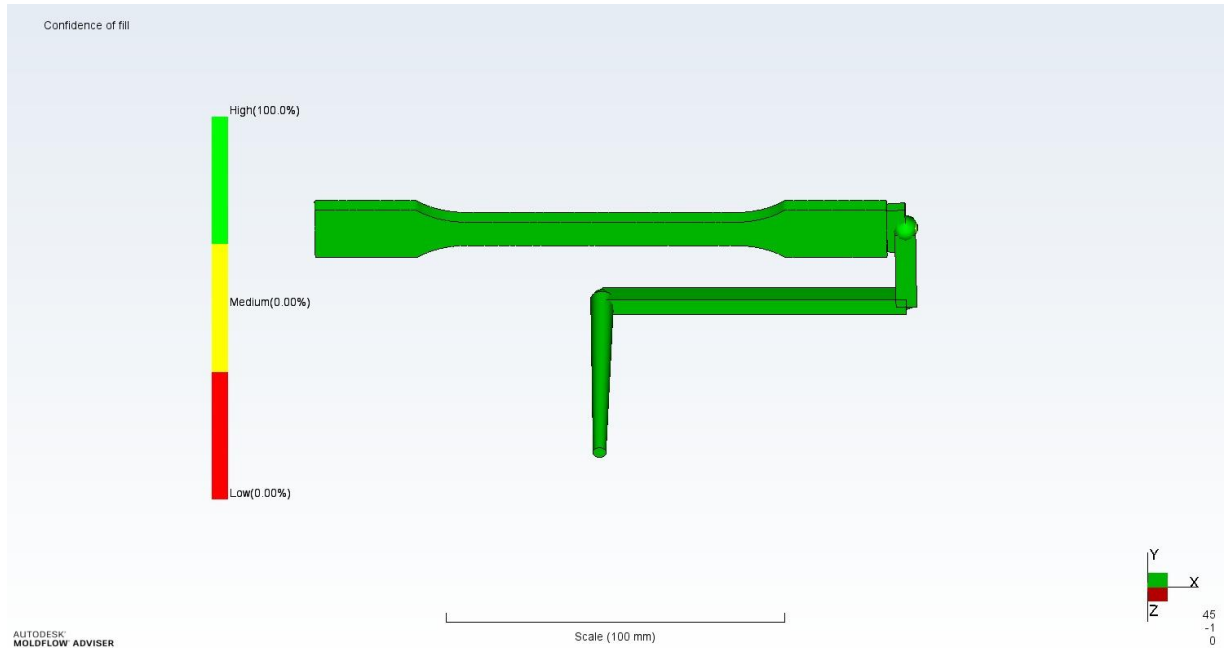


Figure 23 Flow time at different parts of the product.

### 7.4.3. Confidence of Fill

The Confidence of fill results says the probability of plastic filling within the mould cavity under conventional injection molding conditions. These results are derived from the pressure and temperature results.



*Figure 24 Confidence of fill.*

From the above fig... we can see the results displayed in the colors which indicates,

- Green - Definite fill.
- Yellow and red - Maybe difficult to fill or may have quality problems.
- No color - Will not fill and results in a short shot[25].

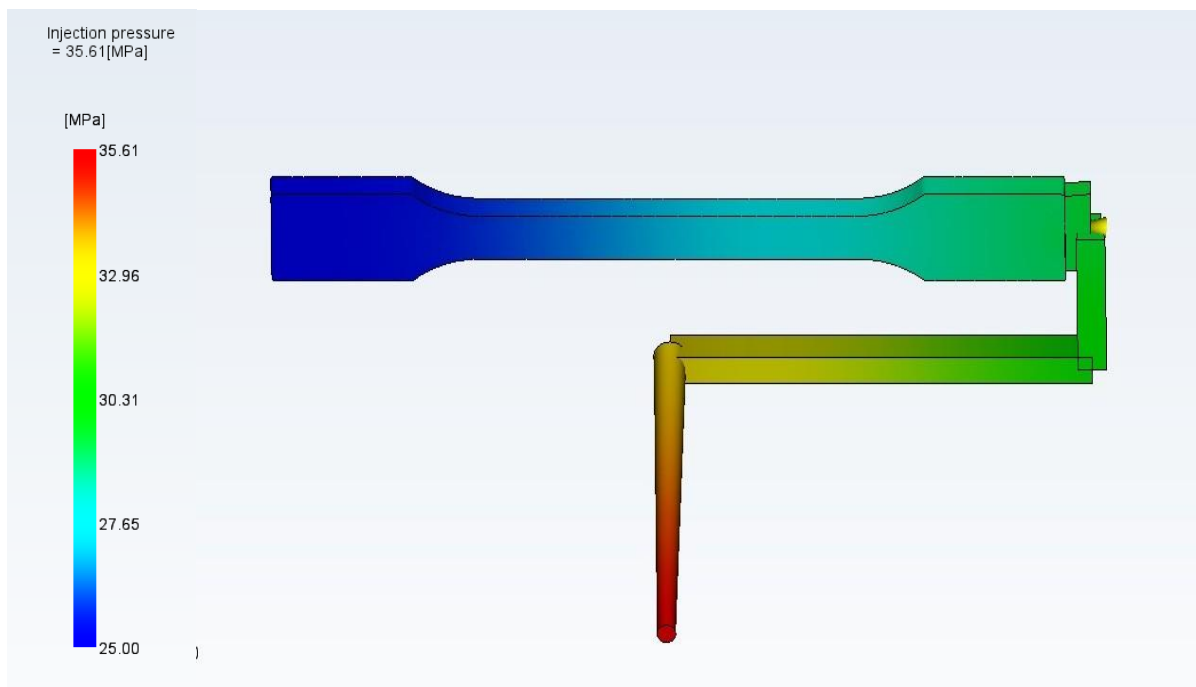
We don't have any problem for this model. We can see the result as 100% confidence of filling all over the part.

#### 7.4.4. InjectionPressure

The pressure variation from one place to another is the force that drives the mold flow during filling. The pressure gradient is the pressure variance divided by the length between two locations.

The injection pressure result gives the most injection pressure value obtained before the velocity or pressure switch-over across during filling. At the start of filling, the pressure is 0 or 1 atm in the absolute pressure scale, throughout the mold. The pressure at a particular location starts to grow only after the melt front approaches that location. It continues to increase as the melt front moves fast, due to the rising flow length within this specific location and the melt front.

Like water flowing from a higher location to lower location, the polymer also does the likewise moves in the direction of the negative pressure gradient, from higher pressure to lower pressure. Therefore the maximum pressure occurs at the injection locations and the minimum happens at the melt front<sup>[25]</sup>. This is shown in the following Fig 21.

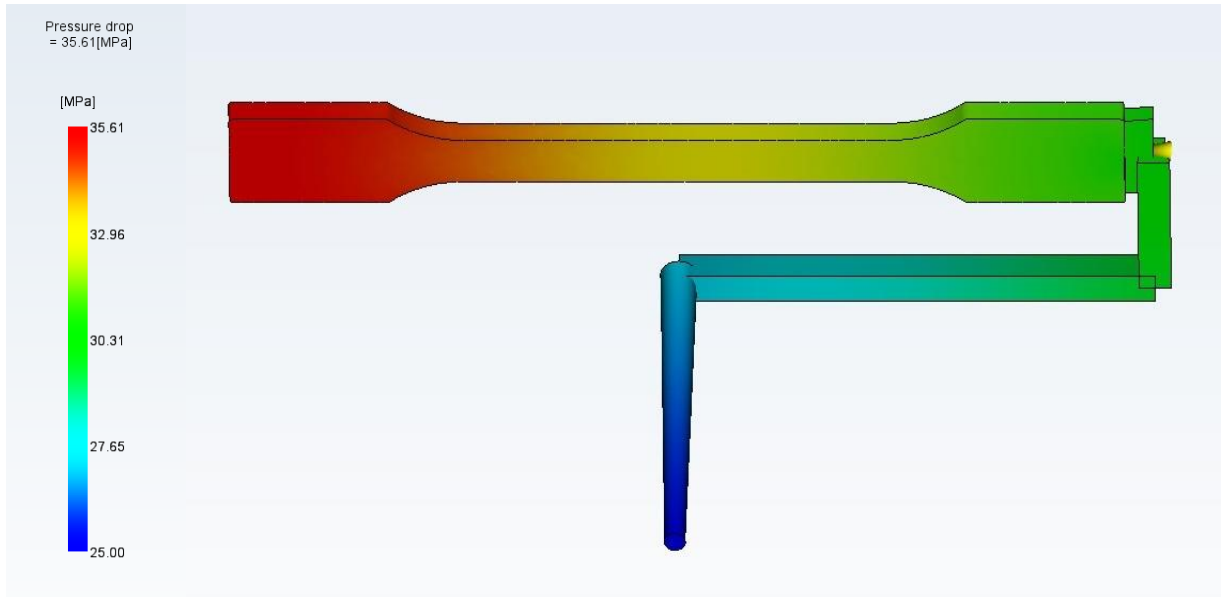


*Figure 25 Injection Pressure.*

From the above Fig.22 we can conclude that the injection pressure of 20 to 30.61 Mpa is used. Also, gives the injection pressure at different locations of the part.

### 7.4.5. PressureDrop

The pressure drop result utilizes different colors to describe the pressure drop in different stages of the filling from high-pressure drop to the lowest pressure drop. This result determines how much pressure is needed to fill the different areas of the part.



*Figure 26 Pressure Drop.*

The pressure drop result is one of the factors that is used to establish the confidence to fill. If the pressure drop is higher than 80% then the confidence of fill will be in yellow, perhaps difficult to fill.

If the pressure drop equals 100% of the current setting, the confidence of fill result for this area is red gives difficulties of fill.

If the default value of 180Mpa for the maximum injection pressure limit is higher than the actual molding machine capacity, the fill result will be better than expected.

The Fig.23 gives the minimum and maximum values of pressure drop which is needed for the process and also shows the drop of pressure at different situations[25].

## 7.4.6. Temperature at Flow front

The Temperature at flow front result provides the temperature of the polymer when the flow front reaches a specified point in the plastic cross-section.

As illustrated in the following diagram, the Temperature at flow front result does have a range of colors to give the region of lowest temperature in blue into the region of the highest temperature in red. The colors symbolize the material temperature at every point as that point was filled. The result exposes the differences in the temperature of the flow front while filling.

The flow front temperature should not drop more than 2°C to 5°C through the filling phase. More principal changes usually indicate that the injection time is too low, or there are areas of uncertainty. If the flow front temperature is too low in a thin area of the part, the delay may result in a short shot. In regions where the flow front temperature rises, material degradation, and surface errors may occur[25].

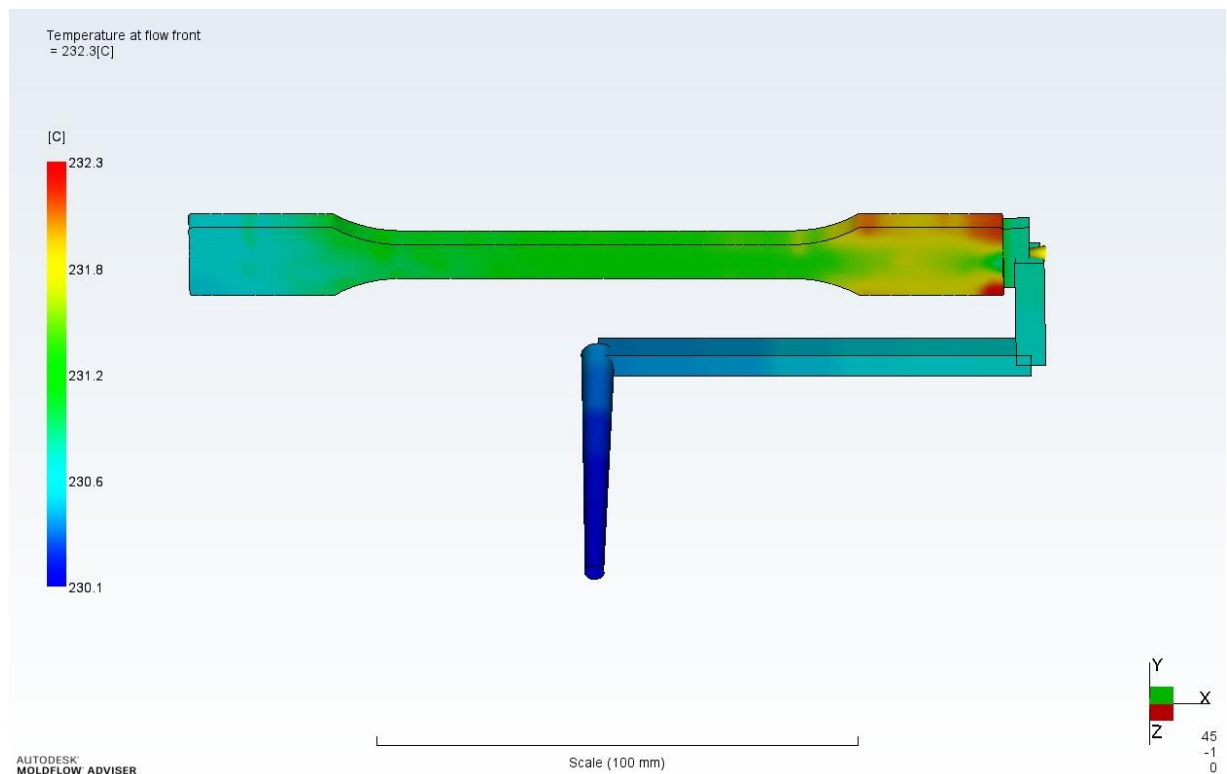
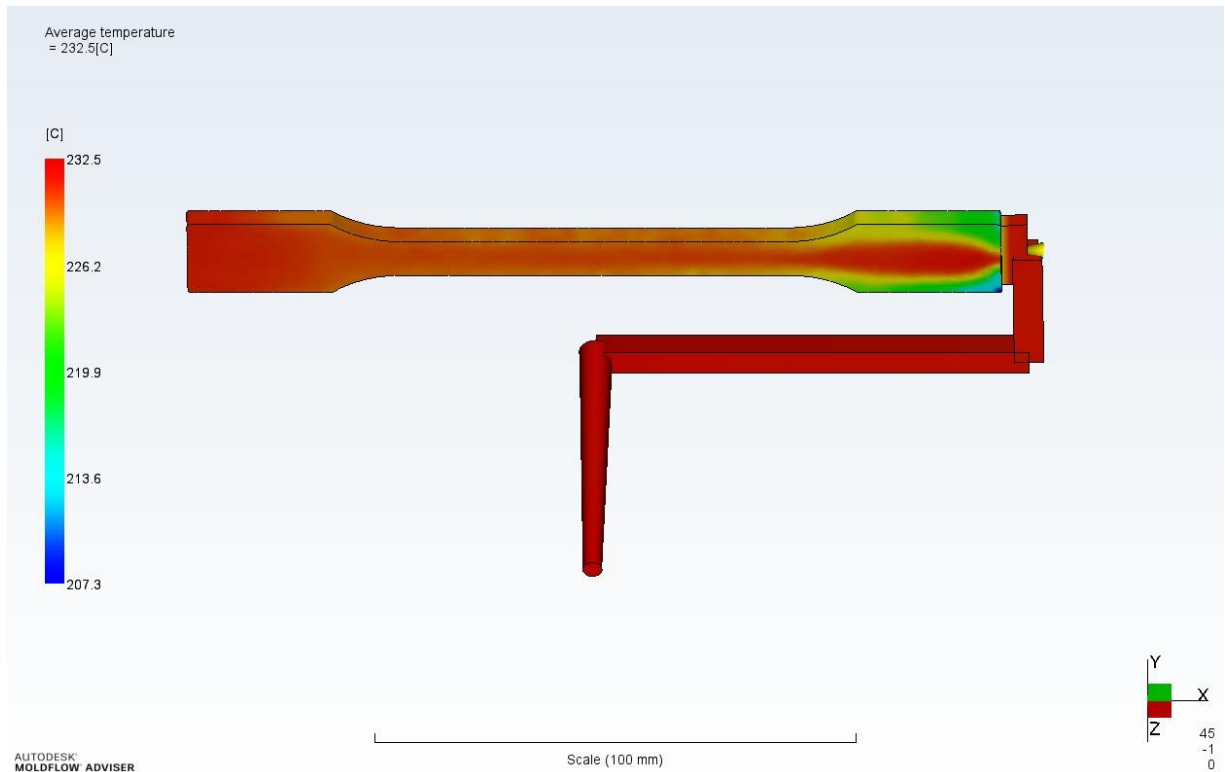


Figure 27 Temperature at flow front.

### 7.4.7. AverageTemperature

The Average temperature result confers the average bulk temperature all over the part by the edge of the fill. The temperature of the melt alters not only with time and the location but also with the thickness during the whole injection cycle. Average temperature represents the energy that is displayed in a specific location at a particular time.

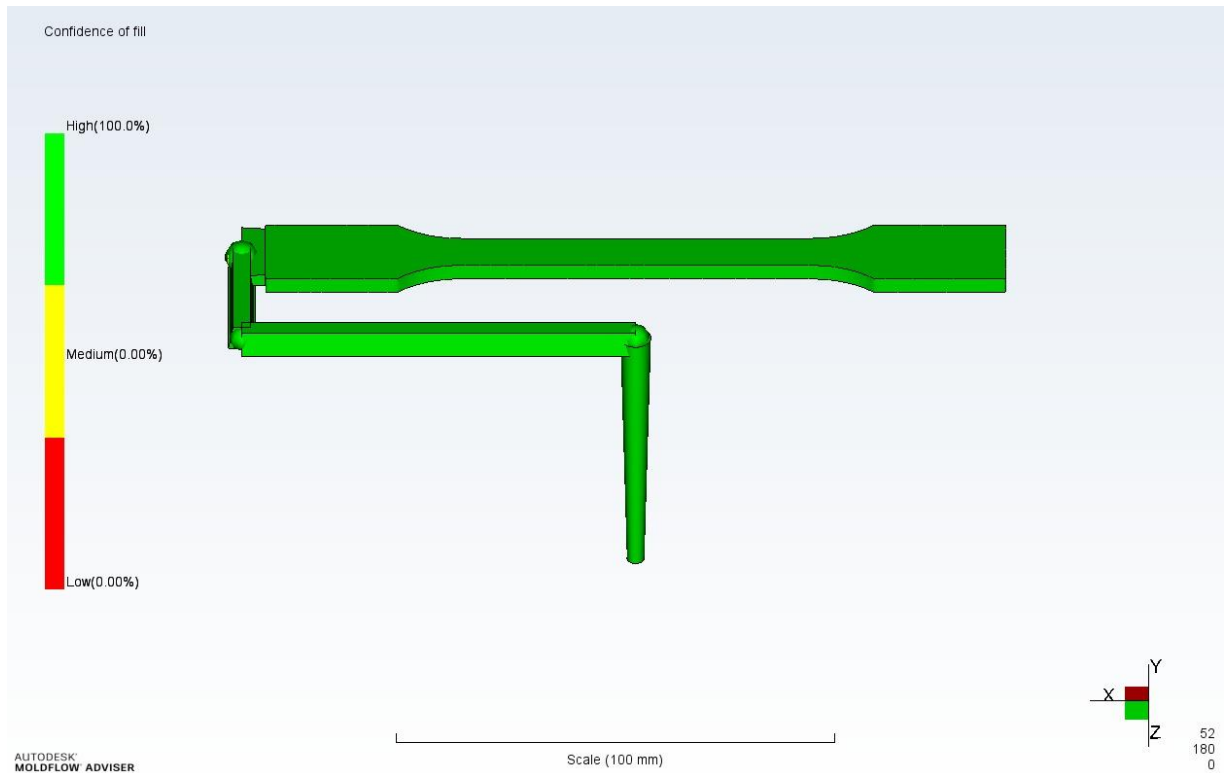


*Figure 28 Average Temperature.*

From the above Fig 24. Areas with steady flow typically have a fabulous average temperature, which falls swiftly when the flow ends. If the average temperature is too light in a thin area of the part, then delaying or short shots happens. Where if it is low at weld lines are present, then the weld lines become more dangerous. If the average temperature is advanced then the degradation happens. To sidestep these difficulties make sure the average temperature is evenly within the suggested limit for the polymer we are practicing[25].

### 7.4.8. QualityPrediction

The Quality result is practiced to show the quality of the mechanical properties and features of the practiced part. It is derived from the pressure, temperature and other results.



*Figure 29 Quality Prediction.*

From the above fig... the green color gradient gives the area of best quality of the product which is 95%. The yellow color section shows that part has quality problems which are 4, 39% where the sprue and the runners are joined.



### 7.4.9 Time to reach ejection temperature

This is given as the time to reach ejection temperature. Where the result shows the amount of time needed to reach the ejection temperature, which is estimated from the start of fill

The result is providing the projected time to freeze if the part has not frozen by the endpoint of the cycle time given.

Generally, the part should freeze evenly. Sections of the part that take longer to freeze may indicate thicker areas of the part or areas of shear heat during filling and/or packing.

Thick areas in the part take a long time to reach ejection temperature, in that case, consider redesigning the part. Long periods are because of shear and these may be difficult to solve. Overcoming the shear may make the Time to reach ejection temperature to resentfully change volumetric shrinkage and warpage [25].

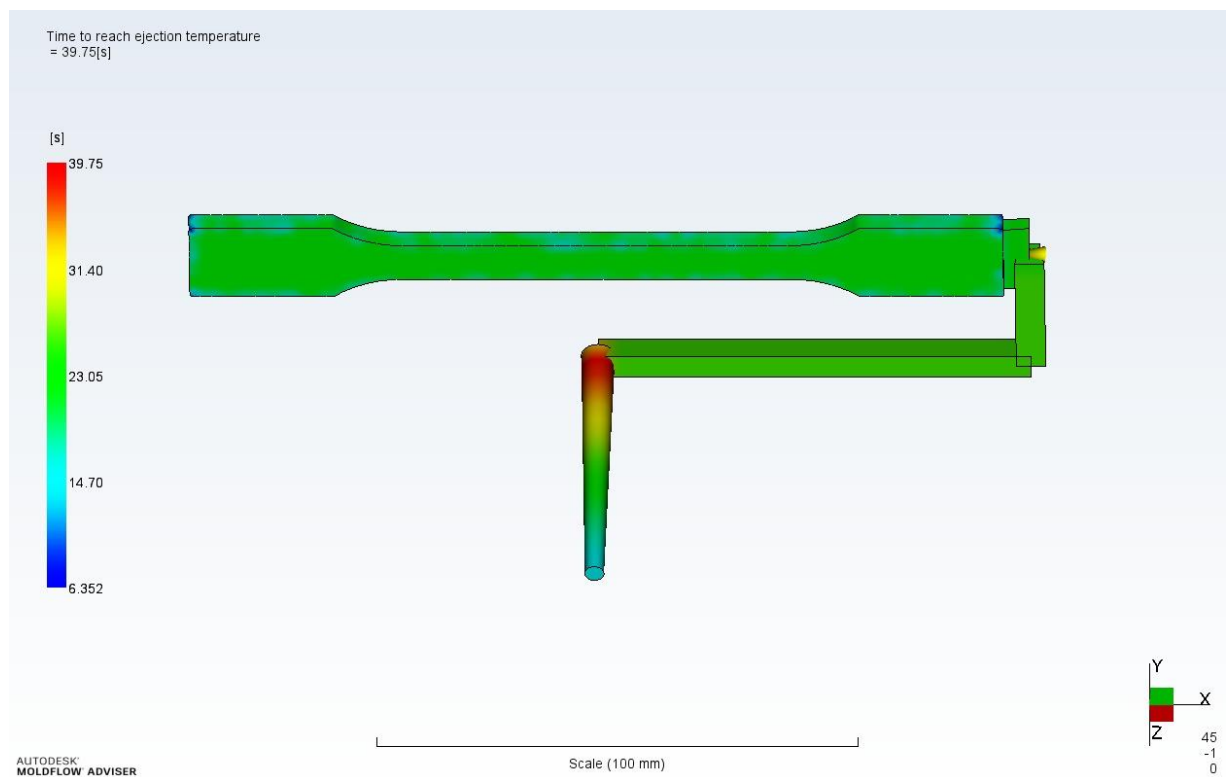


Figure 30 Time to reach ET.

## 8. Task 2

### Understanding the real processing data of the machine

The main intention of this task to analyze and understand the real processing data (Flow time, temperatures and pressures) of the machine which we have in our university.

#### 8.1 Injection moulding Machine

In this experiment, we are going to study the parameters of an Arburg all-rounder 270 C golden edition machine. All-rounder 270c is an injection moulding machine with a horizontally designed injection unit including a clamping force of max 400 KN, stroke volume of max 34 cm, max injection pressure is 2000 bar and the distance between tie bars is 270 X 270 mm. Where the net weight of the machine is 2150 kg.



*Figure 31 Injection moulding machine*

The processing parameters depends on the product to be produced. Where different materials have different properties so we need to set the processing parameters by taking the material characteristics into considerations.

### 8.1.1 Dimensions and setup of the Machine

The dimensions and setup of the Arburg all-rounder 270 C golden edition machine is given in the following Figures.

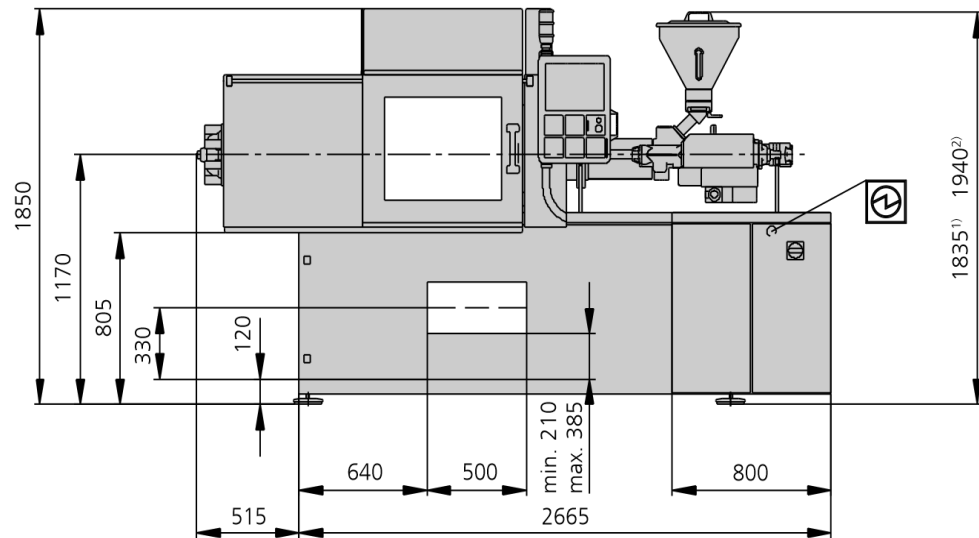


Figure 32 Machine front view[26].

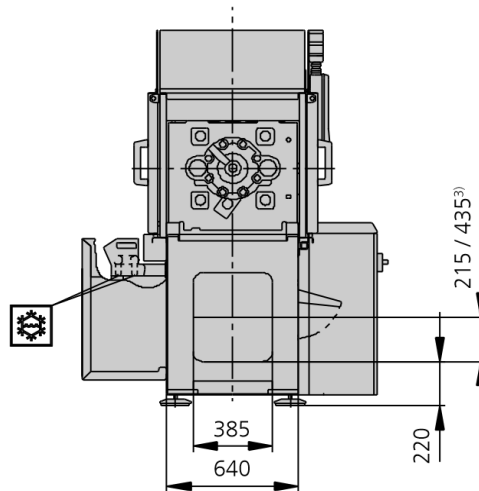


Figure 33 Machine side view[26].

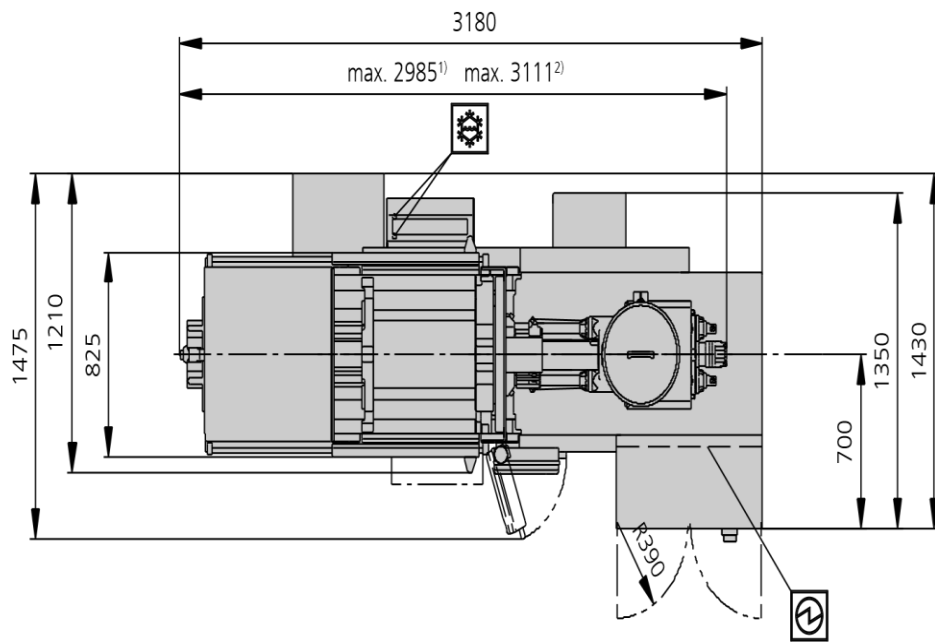
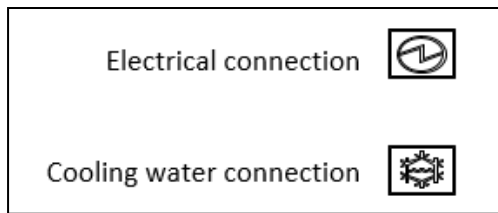


Figure 34 Machine top view[26].



## 8.1.2 Technical data of the machine

The technical data of the machine including injection unit, clamping unit etc., is given in the following tables. The net weight of the machine is 2350 kg. 25L hopper is fixed to the machine for supplying the plastic pellets.

*Table 3 Clamping unit technical data [26].*

<b>Clamping unit</b>	<b>ARBURG ALL- ROUNDER270 C</b>
Clamping force	400 KN
Opening force / Stroke	130 / 350 mm
Mould height	200 mm
Platen daylight	550 mm
Distance between tie bars (w X h)	270 X 270 mm
Mould mounting platens (w X h)	446 X 446 mm
Weight of movable mould half	180 kg
Ejector force / stroke	30 / 125 KN

*Table 4 Technical data of Injection unit[26].*

<b>Injection Unit</b>	<b>ARBURG ALL- ROUNDER270 C</b>
Screw diameter	22 mm.
Effective screw length	20 mm.
Screw stroke	Max. 90 mm
Calculated stroke volume	Max. 34 cm <sup>3</sup>
Injection pressure	Max. 2000 bar
Holding pressure	Max. 2000 bar
Injection flow	Max. 100 cm <sup>3</sup> /s
Screw circumferential speed	Max. 48 m/min
Screw torque	Max 110 Nm
Nozzle contact force / Retraction stroke	50 kN / 150 mm
Heating Capacity / Zones	4.1 / 4 kW

## 8.2 Analyzing the main real processing parameters of the machine

### 8.2.1 Fill time

The time that is needed for the material to fill the cavity. The fill time depends mostly on the temperatures, pressures and materials properties. Injection pressure is the force that drives the material to fill uniformly into the cavity. Material with low liquidity needs more pressure to make it fill all over the mould. Where the shape of the product to be produced decides the time cycle of the moulding process. From the following Fig 33 we can see that the material (polypropylene) needs 0.92 secs for filling all over the product cavity being produced.

We can also see various time profiles from the following fig, which gives the information about holding time, cooling time, and melting time etc. Since, we are discussing about the material filling time I have neglected all the remaining.

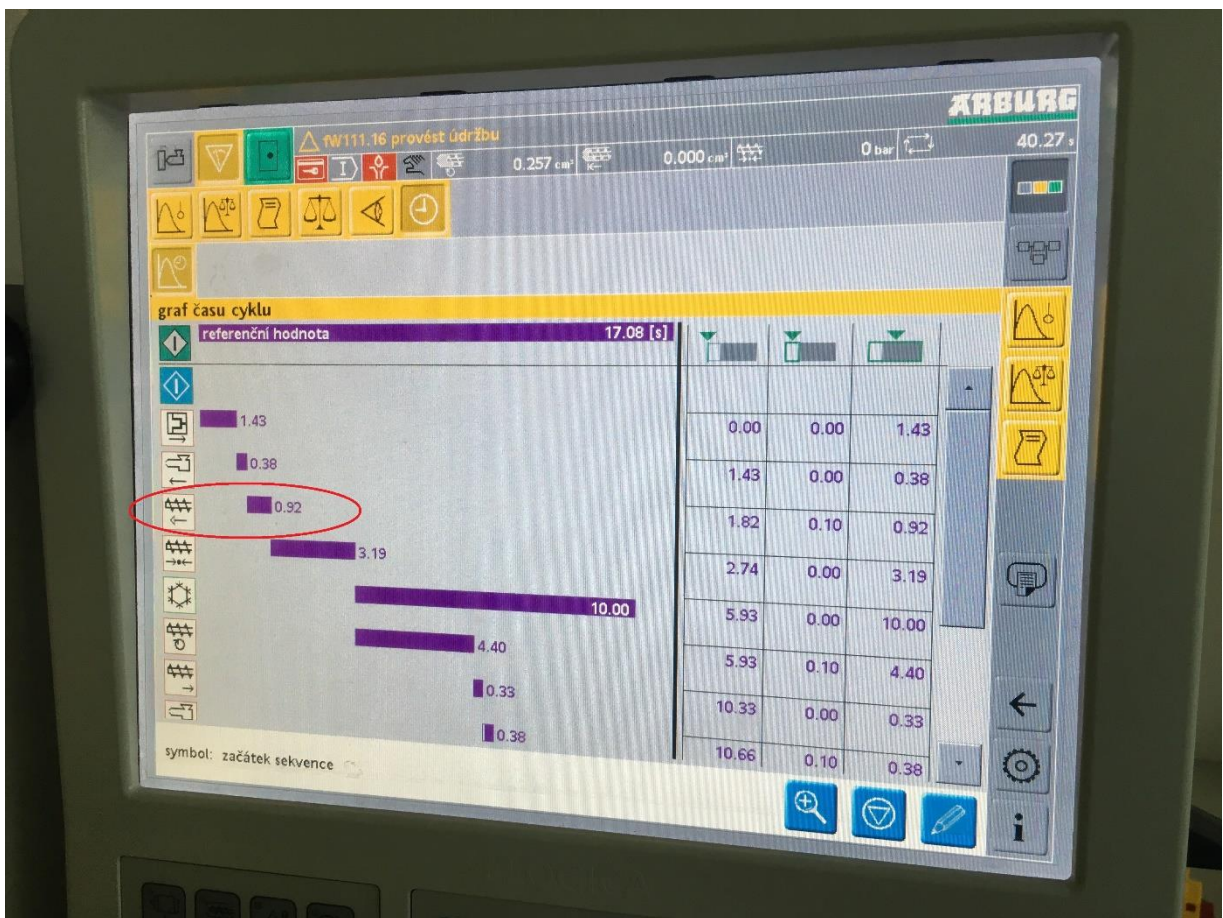


Figure 35 Time profile

## 8.2.2 Injection pressure

Injection pressure usually depends on the material properties, different materials need different values of injection pressures. The difference of the pressure from one location to the other location is the force that makes material to flow while filling. As we can see in the following Fig 34. At first the injection pressure is 300 bar and it is increased later on to 350 bar. Injection pressure depends on density and viscosity of the material as well as temperature. The hotter the barrel, more fluid it makes then the less pressure we need. Where the machine shows that it need 300 to 350 bar to inject the material during filling which is obeying the standard values of the polypropylene material.

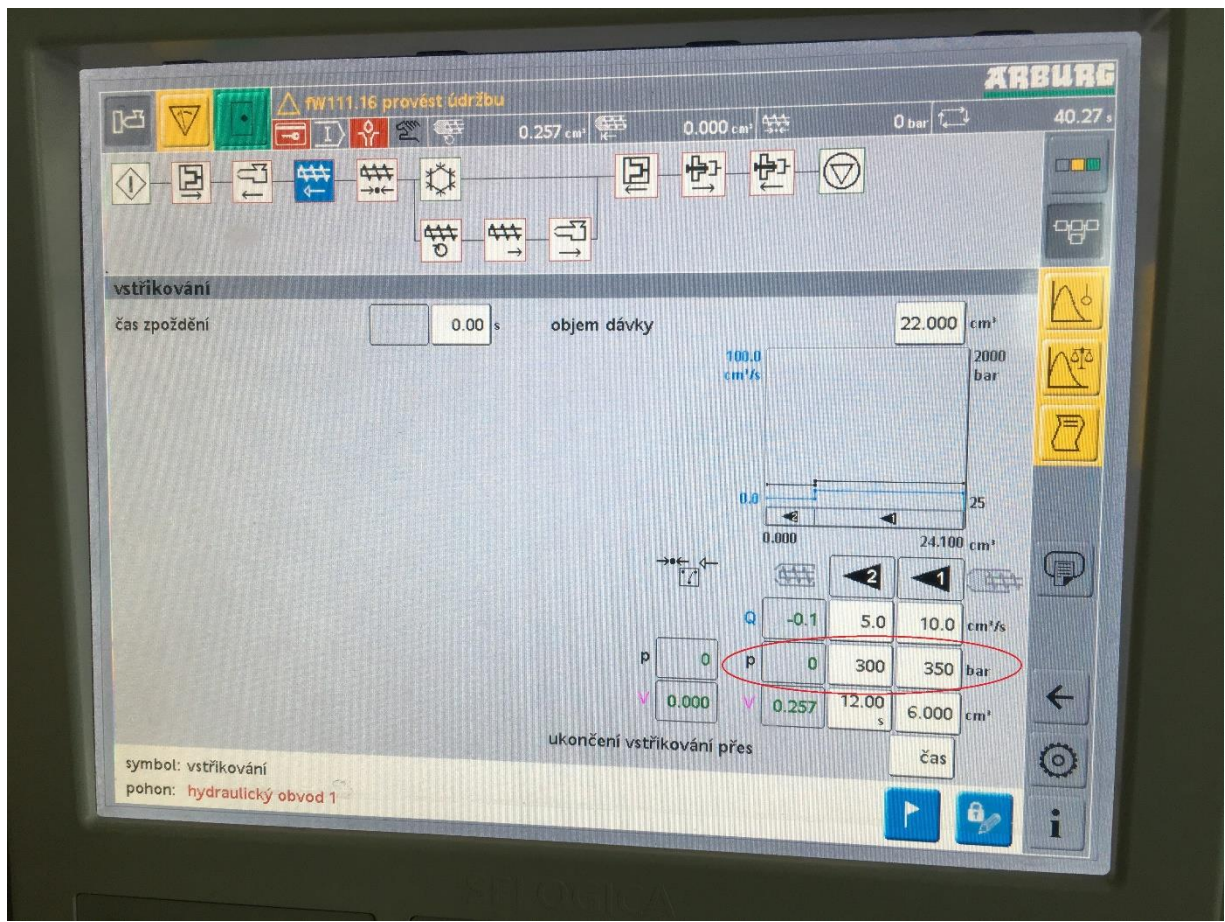


Figure 36 Pressure readings.

## 8.2.3 Temperatures

### 8.2.3.1 Melt temperature

The melt temperature plays the vital role for the flow behavior of the mould, different materials has different values of melting point or temperature which depends mainly on the properties of the material being used for the production. The temperature settings are defined by the material's supplier/seller. Therefore, the machine's temperature settings should set with the material being used for the production. For example, it should be 210° C to 230° C for polypropylene. Since we are using polypropylene, the melt temperature has been already given as 220 degree centigrade as shown in the Fig 35. This fig explains that the material is heated at 220° C by using barrels to melt and flow through the process.

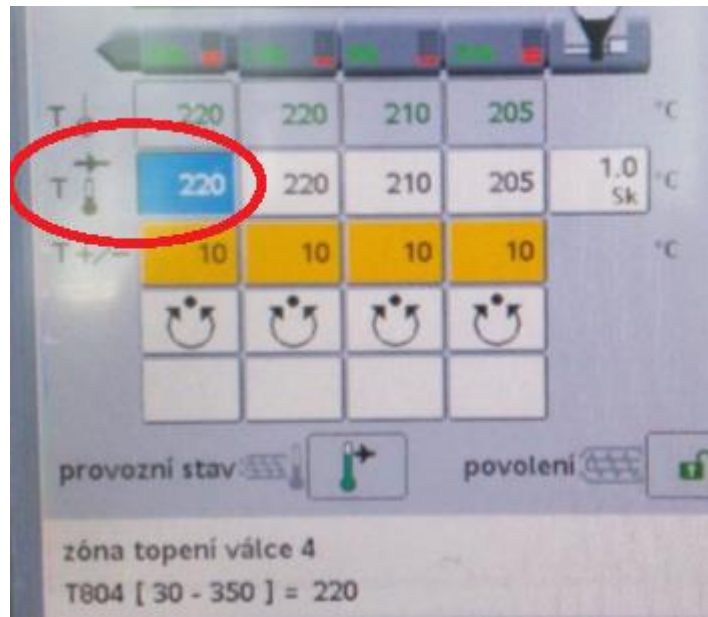


Figure 37 Melt temperature.

### 8.2.3.1 Mould temperature

The temperature in the mould is maintained after material finned completely into the mould cavity. The mold temperature should always be less than the temperature of the barrel which is 220° C where, this will allow the material to cool down. The temperature in the mold is generally between 65 and 130° C. whereas for this product/sample product being produce the mould temperature has been already given in the machine as "80 °C" which is fair enough for polypropylene to cool down in the cavity. Too low temperature can result in knit lines and gives an incomplete part. Too high of a temperature can create warping or blistering in the part. So it is essential to maintain the moderate temperature for better production quality.



## 9. Conclusion:

The main aim of this project is to analyze the processing parameters of the injection molding to meet the requirements for improving the quality of the product. Where the process parameters like temperatures, pressures and flow time play the key role in producing a better product. These process parameters will change according to the material properties to be moulded. Analyzing the material properties and setting the recommended parameters results in a better product.

At first, general information about injection moulding machine and the process of injection moulding was briefly described. The main objective of the literature survey is to study the process and processing parameters of the injection moulding which concludes the product quality. The main theme of the thesis is to perform an experiment by using Software to get the recommended process flow parameters of plastic flow inside the mould cavity and checking recommended software results with the real processing data of the machine. This will give us the best processing parameters to produce an excellent quality product. Consequently, we can optimize the machine data according to the recommended data.

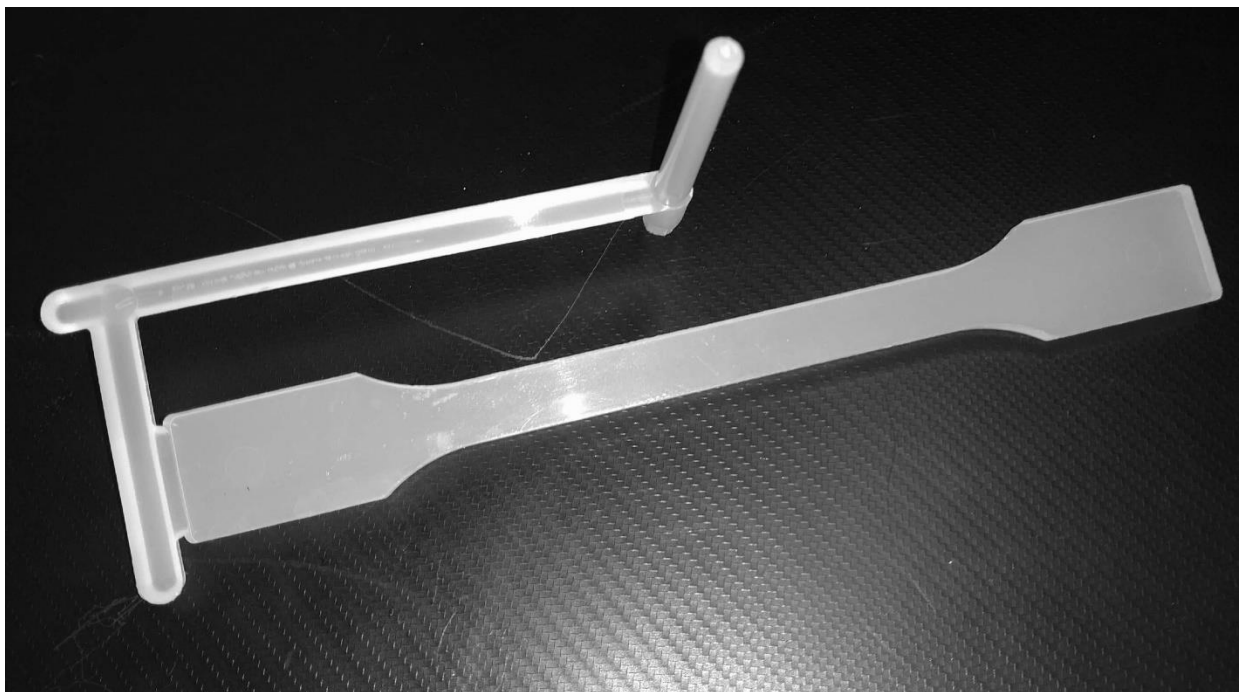
Material and a sample product were selected, designed, imported and simulated by using a software called *Autodesk Moldflow Adviser* which is particularly made for simulating plastic products. The results gave the flow of plastic, flow time, temperatures and pressures needed for the particular material of polypropylene to fill inside the cavity without any problem. Where software also gives the quality profile of the product for the recommended parameters.

The results from the software gave the recommended average melt temperature as 207 °C to 230°C. Where this is the temperature used to heat the material for filling. The suggested injection pressure which drives the material is recommended as 35 MPa at the sprue, runners, and 25 MPa at the cavity. The injection pressure depends mainly on the liquidity of the material. The simulation results also have given the time for filling mould cavity is 1.31 secs and the quality profile says that the final product obtained by using these parameters have 96% better quality and only 4% of quality problems particularly at the runners and sprue.

On the other hand, an experimental task was performed to analyse the flow parameters of the plastic in the machine. For this, I have worked on Arburg all-rounder 270 C, Horizontal injection moulding machine. The temperature that is using by the machine to heat plastic is 220°C. Injection pressure used by the machine for driving the plastic for filling is 300 bar to 350 bar. The results gave the plastic fill time in the cavity as 0.9 secs,

From this, I have concluded that the results which is suggested by software and the practical experiment is almost same. My idea was to compare the software results with the real machine data and to optimize the machine data according to the recommended software results. Unfortunately, the software recommended values are almost same with the machine real process data. Where the machine has already been set with better processing parameters for producing the best quality of the product.

The final product produced from this process is shown in the following Fig.



## Table of Figures

<i>Figure 1 Schematic of a reciprocating screw injection molding machine</i> .....	14
<i>Figure 2 Plasticizing unit</i> .....	15
<i>Figure 3 Clamping unit with a toggle mechanism</i> .....	17
<i>Figure 4 Hydraulic clamping unit</i> .....	18
<i>Figure 5 Injection Mold Cavity</i> .....	19
<i>Figure 6 Schematic of different runner system arrangement</i> .....	20
<i>Figure 7 Schematic of different gating systems</i> .....	21
<i>Figure 8 Barrel Zones</i> .....	24
<i>Figure 9 Injection Molding cycle</i> .....	26
<i>Figure 10 Sequence of events during an injection moulding cycle</i> .....	27
<i>Figure 11 Sample part.</i> .....	28
Figure 12 Model .....	35
Figure 13 Injection location setup .....	36
Figure 14 Gate Setup .....	37
Figure 15 Mold Size.....	38
Figure 16 Runner 1.....	38
Figure 17 Runner 2.....	39
Figure 18 Sprue .....	39
Figure 19 Plastic flow .....	40
Figure 20 Fill time.....	41
Figure 21 Flow time at different parts of the product. ....	42
Figure 22 Confidence of fill. ....	43
Figure 23 Injection Pressure. ....	44
Figure 24 Pressure Drop. ....	45
Figure 25 Temperature at flow front.....	46
Figure 26 Average Temperature.....	47
Figure 27 Quality Prediction. ....	48
Figure 28 Time to reach ET. ....	49
Figure 29 Injection moulding machine .....	50
Figure 30 Machine front view.....	<b>Error! Bookmark not defined.</b>
Figure 31 Machine side view .....	<b>Error! Bookmark not defined.</b>
Figure 32 Machine top view .....	<b>Error! Bookmark not defined.</b>
Figure 33 Time profile.....	54
Figure 34 Pressure readings.....	55
Figure 35 Melt temperature. ....	56

## References

1. Haizolglobal, 2019, Injection molding. [Online]. 2019. [Accessed in Feb 2019]. Available From: [www.haizolglobal.com/capability/molding/injection-molding.shtml](http://www.haizolglobal.com/capability/molding/injection-molding.shtml)
2. Assiplast.com, 2019, what exactly is injection molding. [Online]. [Accessed in Feb 2019]. Available From: <http://www.assiplast.com/exactly-injection-moulding/>
3. Haizolglobal.com, 2019, molding. [Online]. [Accessed in Feb 2019]. Available From: [www.haizolglobal.com/capability/molding/](http://www.haizolglobal.com/capability/molding/)
4. PTFE machinery, 2019, Types of Injection molding machines. [Online]. [Accessed in March 2019]. Available From: [www.nisseiamerica.com/Injection\\_molding\\_machine](http://www.nisseiamerica.com/Injection_molding_machine)
5. Xcentricmold.com, 2019, Injection molding Machine. [Online]. [Accessed in Feb 2019]. Available From: <https://www.xcentricmold.com/wp-content/themes/xcentric/assets/images/about-injection-molding/injection-molding-machine.png>
6. Tim A. Osswald, Lih-Sheng Turng & Paul Gramann, 2007, Injection molding hand book, 2<sup>nd</sup> edition (Topic 1.2.1). [Accessed in Feb 2019].
7. Improve-your-injection-molding.com, 2019, selecting molding screws. [Online]. [Accessed in Feb 2019]. Available From: <https://www.improve-your-injection-molding.com/injection-molding-screws>
8. Improve-your-injection-molding.com, 2019, screw design. [Online]. [Accessed in Feb 2019]. Available From: <https://www.improve-your-injection-molding.com>
9. Tim A. Osswald, Lih-Sheng Turng & Paul Gramann, 2007, Injection molding hand book, 2<sup>nd</sup> edition (Topic 1.12). [Accessed in Feb 2019].
10. taaiglobal.com, 2019, Hydraulic Clamping unit. [Online]. [Accessed in Feb 2019]. Available From: <https://taaiglobal.com/2011/01/11/lorem-ipsam-dolor-sit-amet-consectetur-adipiscing-elit/>
11. www.ecoflow.pl, 2019, Comparison of Toggle and Hydraulic clamping unit. [Online]. [Accessed in March 2019]. Available From: [http://www.ecoflow.pl/files/products/wtryskarki/porownanie\\_ukladu\\_zamykania\\_800.jpg](http://www.ecoflow.pl/files/products/wtryskarki/porownanie_ukladu_zamykania_800.jpg)
12. Tim A. Osswald, Lih-Sheng Turng & Paul Gramann, 2007, Injection molding hand book, 2<sup>nd</sup> edition (Topic 1.2.3, Fig 1.14, 1.15 & 1.16). [Accessed in Feb 2019].
13. www.lanson-imm.com, 2019, the main process parameters of injection moulding machine. [Online]. [Accessed in April 2019]. Available from: <http://www.lanson-imm.com/en/nowledge/71-51.html>
14. plastictroubleshooter.com, 2019, the main process parameters of injection moulding machine. [Online]. [Accessed in April 2019]. Available from: [http://www.plastictroubleshooter.com/ThePlasticTroubleshooter/molding\\_process.htm](http://www.plastictroubleshooter.com/ThePlasticTroubleshooter/molding_process.htm)

15. Azom, 2019, Important of temperature and pressures in moulding process. [Online]. [Accessed in May 2019].  
Available from: <https://www.azom.com/article.aspx?ArticleID=13580>
16. Lanxess, 2019, Barrel zones and temperature. [Online]. [Accessed in March 2019].  
Available from:  
<https://techcenter.lanxess.com/scp/americas/en/techServscp/79183/article.jsp?docId=79173>
17. AccuTherm.com, 2019, Mold Temperature. [Online]. [Accessed in March 2019].  
Available From: <http://accutherm.com/blog/importance-temperature-control-plastic-injection-molding/>
18. Crescent Industries, 2019, Nozzle temperature, important factors of injection moulding. [Online]. [Accessed in May 2019]. Available  
from: <http://info.crescentind.com/blog/bid/59146/5-factors-important-to-injection-molding>
19. Lanson-imm.com, 2019, Time parameter, the main process parameters of injection moulding machine. [Online]. [Accessed in May 2019].  
Available from: <http://www.lanson-imm.com/en/nowledge/71-51.html>
20. Injection molding hand book, 2<sup>nd</sup> edition, by Tim A. Osswald, Lih-Sheng Turng & Paul Gramann. Injection moulding cycle. [Hand book]. [Accessed in Feb 2019].
21. Autodesk, 2019, Auto desk Mold flow advisor software overview [Online]. [Accessed in June 2019].  
Available from: <https://www.autodesk.com/products/moldflow/overview>
22. www.ceresana.com, 2019, everything you need to know polypropylene [Online]. [Accessed in Feb 2019]. Available from:  
[www.ceresana.com/en/market-studies/plastics/polypropylene/](http://www.ceresana.com/en/market-studies/plastics/polypropylene/)
23. www.creativemechanisms.com, 2018, what is polypropylene [Online]. [Accessed in June 2019]. Available from:  
[www.creativemechanisms.com/blog/all-about-polypropylene-pp-plastic](http://www.creativemechanisms.com/blog/all-about-polypropylene-pp-plastic)
24. omnexus.specialchem.com, 2018, Characteristics, advantages & disadvantages of polypropylene [Online]. [Accessed in July 2019]. Available from:  
<https://omnexus.specialchem.com/selection-guide/polypropylene-pp-plastic>
25. Autodesk mold flow, 2019, results summary and overview [software]. [Accessed in May 2019].
26. Arburg, 2019, Injection moulding machine, dimensions and technical data. [Online]. [Accessed in May 2019]. Available from:  
[https://www.arburg.com/fileadmin/redaktion/Mediathek/Technische\\_Daten/ARBURG\\_ALLROUNDER\\_270C\\_GOLDEN\\_EDITION\\_TD\\_523670\\_en\\_GB.pdf](https://www.arburg.com/fileadmin/redaktion/Mediathek/Technische_Daten/ARBURG_ALLROUNDER_270C_GOLDEN_EDITION_TD_523670_en_GB.pdf)