NUMERICAL INVESTIGATIONS ON SPRUE CONFIGURATION FOR MULTI-CAVITY MOULD

ALYA NABILAH BINTI BADRUL HISHAM

UNIVERSITI SAINS MALAYSIA

2022

NUMERICAL INVESTIGATIONS ON SPRUE CONFIGURATION FOR MULTI-CAVITY MOULD

By:

ALYA NABILAH BINTI BADRUL HISHAM

(Matric no: 145036)

Supervisor:

Dr. Mohd Syakirin Bin Rusdi
July 2022

The dissertation is submitted to
Universiti Sains Malaysia
As partial fulfilment of the requirement to graduate with honours degree in
BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

School of Mechanical Engineering
Engineering Campus
Universiti Sains Malaysia

DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed……………………………………………………….. (Alya Nabilah Binti Badrul Hisham)

Date 24 AUGUST ……………………………………………………………………….. (2022)

STATEMENT 1

This thesis results from my own investigations, except where otherwise stated. Other sources are acknowledged by giving explicit references. Bibliography/references are appended.

Signed……………………………………………………….. (Alya Nabilah Binti Badrul Hisham)

Date 24 AUGUST ……………………………………………………………………….. (2022)

STATEMENT 2

I give consent for my thesis, if accepted, to be available for photocopying and for interlibrary loan, and for the title and summary to be made available outside organisations.

Signed……………………………………………………….. (Alya Nabilah Binti Badrul Hisham)

Date 24 AUGUST ……………………………………………………………………….. (2022)

ACKNOWLEDGEMENT

This Final Year Project would not have been possible without the exceptional support of my supervisor, Dr. Mohd Syakirin Bin Rusdi. His guidance, assistance, and patience have encouraged to complete this project despite having setbacks along the way. I would like to also express my gratitude to my fellow supervisee, Muhammad Zulhilmi Bin Mohd Zaki, who always gladly provide helpful insights and taught me computational fluid dynamics. I am ever so grateful to have both of them throughout during the completion of this project. I am also thankful to En Jamari Sadli who have worked hard to maintain the CAD laboratory to ensure the PCs are operative for all students and allow this project and many more to complete. I would also like to thank my parents and siblings who never hesitated to support me in more ways that I could imagine. My family have also been the most significant to me in the pursuit of this project than anyone else. They are the ultimate sources of inspiration. Most importantly, I want to express my gratitude to my kind and encouraging friends who never stop inspiring me and always ready to lend a helping hand. They are indeed a friend in need and always around to motivate and assist me personally and academically.

TABLE OF CONTENTS

DECLARATION ii

ACKNOWLEDGEMENT iii

TABLE OF CONTENTS iv

LIST OF FIGURES vii

LIST OF GRAPHS viii

LIST OF TABLES ix

LIST OF ABBREVIATIONS x

ABSTRAK xi

ABSTRACT xii

CHAPTER 1 INTRODUCTION 1

1.1 General Introduction 1

1.2 Problem Statement 4

1.3 Objectives 4

1.4 Scope of Work 5

1.5 Thesis Outline 5

CHAPTER 2 LITERATURE REVIEW 6

CHAPTER 3 RESEARCH METHODOLOGY 9

3.1 Introduction 9

3.2 Numerical Method 9

3.2.1 Navier-Stokes Equation 9

3.2.2 Generalized Newtonian Fluid (Power Law Fluid) 10

3.2.3 Viscosity Model (Cross-WLF Model) 11

3.2.4 Volume of Fluid (VOF) Method 12

3.3 Designing and Modelling in Solidworks 13

3.3.1 Design Considerations 13

3.3.2 Sprue Design 17
3.4 Simulation Setup in ANSYS Fluent 19
3.4.1 Mesh development in ANSYS Mechanical 20
3.4.2 Boundary Condition Definition 20
3.5 Material, Solver and Phases 22
3.6 Data Collection 23
3.7 Simulation Accuracy 24
3.7.1 Time Step Study 24
3.7.2 Grid Dependency Test 25
CHAPTER 4 RESULTS AND DISCUSSION 28
4.1 Effect of PP Injection Pressure 28
4.1.1 PP Pressure vs Flow Time 29
4.1.2 PP Velocity vs Flow Time 31
4.1.3 PP Fill Time vs PP Injection Pressure 32
4.1.4 PP Velocity vs PP Injection Pressure 33
4.2 Effect of Different PP Temperature 34
4.2.1 PP Pressure vs Flow Time 35
4.2.2 PP Velocity vs Flow Time 37
4.2.3 PP Fill Time vs PP Injection Temperature 38
4.2.4 PP Velocity vs PP Temperature 39
4.3 Effect of PP Grade 40
4.3.1 PP Outlet Pressure vs Flow Time 40
4.3.2 PP Velocity vs Flow Time 42
4.3.3 PP Fill Time vs PP Grade 43
4.4 Summary 43
CHAPTER 5 CONCLUSION AND FUTURE RECOMMENDATIONS 45
5.1 Conclusion 45

5.2 Future Recommendations 47

REFERENCES 48

APPENDIX 51

LIST OF FIGURES

Figure 3.1 Sprue outlet orifice equation (Engelmann & Dealey, 1999) 13

Figure 3.2 Types of runner designs (Hatch, 2003) 14

Figure 3.3 Types of cold slug well designs (B.-K. Lee, n.d.) 16

Figure 3.4 Sprue anatomy 17

Figure 3.5 Sprue dimensions (Polyplastics, 2012).. 17

Figure 3.6 3D model of sprue 18

Figure 3.7 Sprue dimensions.. 18

Figure 3.8 Simulation process flowchart 19

Figure 3.9 Meshing of the sprue 20

Figure 3.10 The gate system boundaries.. 21

Figure 3.11 Boundaries of the sprue 21

LIST OF GRAPHS

Graph 3.1 LB6331 Polypropylene Cross Model....23
Graph 3.2 TP340 Polypropylene Cross Model....23
Graph 3.3 Runner Pressure vs Time Step....25
Graph 3.4 Runner Pressure vs Flow Time....27
Graph 4.1 PP Runner Average Pressure vs Fill Time for Different PP Injection Pressure....29
Graph 4.2 PP Well Average Pressure vs Fill Time for Different PP Injection Pressure....29
Graph 4.3 PP Average Velocity vs Flow Time for Different PP Injection Pressure....31
Graph 4.4 PP Fill Time vs PP Injection Pressure....32
Graph 4.5 PP Velocity vs PP Injection Pressure....33
Graph 4.6 PP Runner Average Pressure vs Flow Time for Different PP Temperature....35
Graph 4.7 PP Well Average Pressure vs Flow Time for Different PP Temperature....35
Graph 4.8 PP Well Average Velocity vs Flow Time for Different PP Temperature....37
Graph 4.9 PP Fill Time vs PP Temperature....38
Graph 4.10 Maximum Runner Velocity vs PP Temperature....39
Graph 4.11 PP Runner Pressure vs Flow Time for Different PP Grade....40
Graph 4.12 PP Well Pressure vs Flow Time for Different PP Grade....41
Graph 4.13 PP Well Velocity vs Flow Time for Different PP Grade....42

LIST OF TABLES

Table 3.1 Typical runner diameters for unfilled generic materials (Zhilian Mould, 2012) 15

Table 3.2 Sprue dimensions 17

Table 3.3 Boundary and Initial Conditions 22

Table 3.4 LB6331 rheology data 22

Table 3.5 TP340 rheology data 23

Table 3.6 Time step study results 24

Table 3.7 Number of elements according to mesh size 26

Table 3.8 Grid independence test results 26

Table 4.1 Effect of Different PP Injection Pressure 28

Table 4.3 Effect of Different PP Temperature 34

Table 4.5 Effect of Different PP Temperature 40

Table 4.7 PP Fill Time vs PP Grade 43

Table 4.8 Summary of effect of PP injection pressure, temperature, and grade 43

Table 5.1 Results for Different PP Injection Pressure 51

Table 5.2 Results for Effect of Different PP Temperature 52

Table 5.3 Results for Effect of Different PP Temperature 53

LIST OF ABBREVIATIONS

CAE	Computer Aided Engineering
CAD	Computer Aided Design
CFD	Computational Fluid Dynamic
PP	Polypropylene
ASA	Acrylonitrile Styrene Acrylate
WLF	Willam-Landel-Ferry
NS	Navier-Stokes
GNF	Generalized Newtonian Fluid
VOF	Volume of Fluid
RSM	Response Surface Methodology

ABSTRAK

Polipropilena ialah polimer tambahan termoplastik yang terdiri daripada beberapa monomer propilena. yang sering digunakan secara komersial kerana sifatnya yang serba boleh. Kebiasaanya, produk PP dihasilkan melalui pengacuan suntikan dan penyemperitan. Pengacuan suntikan ialah teknik pembuatan dimana cecair disuntik ke dalam acuan. Matlamat keseluruhan kajian adalah untuk lebih memahami bagaimana PP mengalir semasa pengacuan suntikan. dengan mereka bentuk konfigurasi sprue untuk acuan berbilang rongga, menjalankan penyiasatan berangka pada konfigurasi untuk mencari tekanan dalam pelari, dan halaju bendalir semasa suntikan serta membandingkan kebolehliran antara PP gred komersial dan PP gred perubatan. Penyiasatan dilakukan dengan menjalankan simulasi berangka menggunakan Ansys Fluent 2020 R1, dirangkai dalam Ansys Mechanical 2021 dan dimodelkan dalam SolidWorks 2020. Hasil simulasi menunjukkan bahawa tekanan suntikan yang lebih tinggi (49 MPa) menghasilkan tekanan pelari yang lebih tinggi (26466.66 KPa) serta halaju bendalir (256.06 m/s) di mana masa pengisiannya pendek (0.91615 s) berbanding tekanan suntikan yang lebih rendah (45). MPa). Sebaliknya, suhu suntikan yang lebih tinggi (220°C) menghasilkan tekanan pelari (27973.42 KPa) tetapi halaju bendalir yang lebih rendah (233.34 m/s) di mana masa pengisiannya pendek (0.90745 s) berbanding suhu suntikan yang lebih rendah (180°C). Fenomena ini mungkin berlaku disebabkan oleh penipisan ricih yang mempengaruhi kelikatan cecair bukan Newtonian di mana kelikatan berkurangan di bawah terikan ricih. Gred perubatan PP LB6331 mempunyai kebolehliliran yang lebih tinggi daripada gred komersial PP TP6331 kerana menunjukkan tekanan pelari yang lebih tinggi (26888.08 KPa), halaju bendalir yang lebih tinggi (257.02 m/s), dan masa pengisian yang lebih rendah (0.93265 s).

ABSTRACT

Polypropylene is a thermoplastic addition polymer which is made up of several propylene monomers. PP is a popular material used commercially as it is a versatile material. Manufacturing of PP products through injection moulding and extrusion is common. Injection moulding is a manufacturing technique that enables large-scale production of items which includes injecting molten materials into a mould. The overall goal of the study is to better understand how PP flows during injection moulding. The first goal is to design a sprue configuration for multi-cavity mould. The second objective is to conduct numerical investigations on the configuration to find the pressure in the runners, and fluid velocity during injection. The final objective is to compare the flowability between commercial grade PP and medical grade PP. The investigation is done by conducting a numerical simulation, which is carried out using Ansys Fluent 2020 R1, while the sprues are meshed in Ansys Mechanical 2021 and modelled in SolidWorks 2020. The simulation results show that higher injection pressure (49 MPa) results in higher runner pressure (26466.66 KPa) as well as the fluid velocity (256.06 m/s) which gives lower fill time (0.91615 s) as opposed to lower injection pressure (45 MPa). On the other hand, higher injection temperature (220°C) results in runner pressure (27973.42 KPa) but lower fluid velocity (233.34 m/s) which gives lower fill time (0.90745 s) as opposed to lower injection temperature (180°C). This phenomenon may occur due to shear thinning which influence the viscosity of a non-Newtonian fluid where the viscosity decreases under shear strain Other than that, the medical grade PP LB6331 generally has a higher flowability than the commercial grade PP TP6331 due to exhibiting higher runner pressure (26888.08 KPa), higher fluid velocity (257.02 m/s), and lower fill time (0.93265 s).

CHAPTER 1

INTRODUCTION

1.1 General Introduction

Injection moulding is a manufacturing technique that enables large-scale production of items. It works by injecting molten materials into a mould. It's usually used to make thousands of identical things as part of a large production process. Metals, glasses, elastomers, and confections are among the materials used in injection moulding, but thermoplastic and thermosetting polymers are the most prevalent.

In injection moulding, moulds are mostly made from metal and machined with high precision to replicate the features of the part that will be manufactured. The mould will then be fitted into the injection moulding machine where molten material will be injected to fill its cavity completely, then cooled until it solidifies. Before the material is injected into the mould, the material is first fed into a heated barrel which has gradual change of temperature along the barrel. The material also gets mixed along the barrel with a helical screw. Finally, when the material has completely solidified, the mould is opened, and the finished product is retrieved.

Moulds can be divided into two: Single cavity or multi cavity. A single-cavity mould can produce one part at a time, while the multi cavity mould is a mould which can produce multiple parts at a time. A multi cavity mould can produce identical parts or unidentical parts. Aluminium and steel moulds are typically used in injection moulding. A two-shot mould is a type of injection moulding that allows different materials to be blended in one part. This technique can be used to provide plastic products a softer feel, to colour a part, or to create objects with varied performance characteristics.

When to it comes to mould material, aluminium moulds are more economic than steel moulds, however, is not as durable as steel moulds. Thus, aluminium moulds are not recommended for high volume manufacturing or products with tight dimensional tolerances. This is because aluminium has poor mechanical qualities and are prone to wear, deformation, and breakage sustained from forces which includes clamping and injection forces.

As for the moulding material, polypropylene (PP) is a popular choice. Polypropylene is thermoplastic addition polymer which is made up of several propylene monomers. PP is a versatile material, and its application includes commercial product packaging, or becomes the preferred material for plastic components such as in the automotive industry. PP is translucent, semi rigid, robust and is fatigue resistant, heat resistant and chemically resistant. Manufacturing of PP products through injection molding and extrusion is common.

PP is often favored in mass production as it is economic, lightweight and is easy to work with in injection molding despite being semicrystalline. From the standpoint of mechanical properties, the lack of a true necessity for high molecular weight results in low melt viscosity thus possesses easy flow. At high shear rates, the pseudoplastic structure of polypropylene accentuates this effect thus has fast filling rates.

On the other hand, when it comes to mould design, there are numerous things to be considered when designing a mould. This includes the cost of the mould design, the volume of production, defect prevention, etc. With advancements in computer technology and artificial intelligence, efforts have been made to lower the cost and lead time associated with designing and manufacturing an injection mould. Because injection mould design is a complex process including multiple sub-designs relating to various components of the mould, each requiring professional knowledge and expertise, it has been the focus of research.

Mould design has an impact on productivity, mould maintenance costs, mould manufacturability, and moulded part quality. The majority of mould design research has focused on using expert systems, knowledge-based systems, and artificial intelligence to replace or enhance the large amount of human skill necessary in the traditional design process.

1.2 Problem Statement

In multi-cavity injection moulding, there are several injection parameters needed to be considered which includes the injection pressure, temperature and material. These injection parameters can determine the quality of the products as well as the cost and timing to produce a product. By using CAD and CFD Simulation, the flow of the injection can be predicted before the actual manufacturing process. In this project, we are focusing on exploring effects of different parameters to injection moulding and how they affect the performance of mould filling. This is done by using CAD and CFD simulation to design and to study the flowability of the molten thermoplastic in the sprue design to find the injection pressure, temperature, and fluid velocity.

1.3 Objectives

The overall goal of the study is to better understand how PP flows as it is being filled into a multi-cavity mould sprue by means of three-dimensional numerical simulation. The following primary goals were established:

1. Design a sprue configuration for multi-cavity mould with special consideration of the intersection between the sprue, runner, and cold slug well.
2. Conduct numerical investigations on the configuration to study the flowability of the molten thermoplastic in the sprue designs to find:
   a. the pressure in the runners, and
   b. fluid velocity during injection.
3. Compare the flowability between commercial grade PP and medical grade PP.

1.4 Scope of Work

The sprue configuration and cavity configuration design will be designed in SolidWorks. The design done on SolidWorks will then be exported into ANSYS Fluent 2021 where the simulation of the part will be conducted for the numerical investigation purposes. In ANSYS, the part will undergo meshing, and the determination of the parameters of the sprue, input and material of the molten plastic that will be injected in the mould.

1.5 Thesis Outline

This thesis is comprised of five chapters. The first chapter, Chapter 1 introduces generally about injection moulding, specifically about thermoplastic injection moulding, and numerical simulation. Chapter 1 also explains the problem statement, the objectives, and the scope of this study. Chapter 2 covers the literature review on numerical simulation of the injection moulding process. Other than that, Chapter 3 explains the numerical simulation method, the design process as well as the simulation setup and accuracy tests conducted prior to the simulation. On the other hand, Chapter 4 describes and discusses the results of the study according to the objectives of the study. Finally, Chapter 5 concludes the findings of this study and shares some recommendation for future work.

CHAPTER 2

LITERATURE REVIEW

Injection moulding is a process where the moulding material melts inside a heating cylinder and injects it into the mould tool, where it solidifies to create the moulded product. The injection moulding machine on the other hand comprises of mould clamping device opens and closes the mould tool, and a device plasticizes and injects the moulding material (Mitsubishi Engineering-Plastics Corporation, 2011). Injection mould design for polymers is a massive undertaking for a designer in modern environment. It is necessary to create a tool that can make a high number of parts in a short amount of time while consuming the least amount of material, time, energy, and money. As a result, tools must be able to withstand high temperatures and enormous loads, and products must be useful and of excellent quality (Imamovic et al., 2020). It is necessary to developing a quality mould that will allow it to work efficiently at the specified cycle time, allowing for the processing to be completed with few faults during mass production. A multi cavity mould creates multiple products of the same form in a symmetrical layout (Dahan et al., 2012).

Optimisation of the multi cavity injection mould can greatly impact in manufacturing performance as well as make the whole process more economic. Huang et. al. (2009) studies the optimized design of cavity layout and feed system of multi-cavity injection mould. The study first investigates the cavity layout, following to the feed system and design of the mould structure. As for the cavity layout, there are two proposed layouts: Either the plastic parts are either arranged in a rectangular manner or arranged along the longitudinal centerline of the injection mould. The later is considered to be more favourable as there are a lot of space wasted in the former arrangement. Onto the design of the scheme of feed system, several considerations were taken into account such as the design of sprue and gate and the runner system. The proposed sprue dimensions feature a 3.5mm diameter at its tip with a 2° taper. Neither the length of the sprue nor length of cold slug well has been mentioned in the study. Next, the design of the runner system investigates non-balanced and balanced layout of runner system. It is thought that a balanced layout can allow the plastic melt to fill all of the cavities at the same time which was not achieved by the non-balanced layout, and the resulting part has consistent mechanical properties. However, Lakkanna et. al. (Lakkanna et al., 2016) has proposed a design for sprue without the cold well slug complete with a detail drawing for reference.

Computational fluid dynamics (CFD) is a branch of research concerned with the development of numerical solutions for the system of coupled partial differential equations. CFD is under the umbrella term Computer Aided Engineering (CAE). The finite-volume approach is the most widely used numerical methodology for the regime of incompressible, viscous flow, including multi-component gas mixtures. It entails discretizing the governing equations on structured or unstructured computing grids and solving the resulting system of algebraic equations in an iterative manner using a pressure-correction scheme (Razeghi & Henini, 2005).

Huang et. al. (Huang et al., 2009) utilises Moldflow software to conduct CAE simulation analysis to simulate material flow in multi cavity. In the study, several analyses are conducted using numerical modelling including fill time analysis and pressure at injection location analysis. Imamoviz et. al. (Imamovic et al., 2020) also utilised Moldflow to study the injection moulding tools of ASA polymers alongside CATIA and VISI Vero software for tool synthesis. The study compares the parameters numerically and analytically. The parameters that was compared include cooling time, ejection temperature. Other than that, Hamsin et. al. (Hamsin et al., 2009) also uses Moldflow software for mould design exercise instead of through trial-and-error technique that was used prior. The study requires a viscosity models for flow analysis function to simulate the injection mould runner. There are several well-known models available, including the Power law model, the Carreau model, the Cross model, and the Ellis model. The goal of using a viscosity model is to replicate the material's observed behaviour as precisely as feasible. On the other hand, Dahan et. al. (Dahan et al., 2012) utilizes of Cadmould to simulate the melt flow of multi cavity moulds to study ITSB in multi cavity family injection mould. In the study, the main variables used are the temperature distribution, the melt filling and volume shrinkage. Pinarbasi et. al. (Pinarbasi et al., 2003) uses the I-DEAS software to simulate the polymer melt flow control for a multi cavity mould. The purpose of this study was to numerically model and validate a physical capability to adjust the filling of specific cavities during injection moulding of multi-impression tooling products. Lastly, Rusdi et. al. (Rusdi et al., 2016) employs ANSYS FLUENT 14 to investigate injection moulding process numerically. In the study, several variables are considered including the operating temperature and the setup pressure.

CHAPTER 3

RESEARCH METHODOLOGY

3.1 Introduction

The investigation is done by conducting a numerical simulation, a simulation involving several mathematical models to solve the motion of the fluid and simulate the behaviour of the injection moulding process. The simulation is carried out using a CFD software Ansys Fluent 2020 R1, while the sprues are meshed in Ansys Mechanical 2021 and modelled in SolidWorks 2020.

3.2 Numerical Method

3.2.1 Navier-Stokes Equation

The Navier-Stokes equations offer the most direct model are considered to be the governing differential equations of motion of a viscous incompressible fluid motion. The equation is in accordance with the laws of conservation of mass and momentum, and Stokes's hypothesis (Heywood, 2006). The conservation of mass (continuity equation) is given by:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{3.1}$$

The energy equation is given by:

$$\rho c_p\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right)+\eta\dot{\gamma} \tag{3.2}$$

Where,

k	thermal conductivity
T	temperature,
η	viscosity
$\dot{\gamma}$	shear rate

There are several assumptions made for model by which the fluid:

- Is a Generalized Newtonian Fluid (GNF).
- Has constant density.
- Undergoes a non-isothermal process.
- Has a three-dimensional, laminar, and incompressible flow.

3.2.2 Generalized Newtonian Fluid (Power Law Fluid)

A generalized Newtonian fluid is an idealized fluid which the shear stress is dependent on the shear rate, but independent of time. The thermoplastic, polypropylene (PP), is assumed to be a Generalized Newtonian Fluid (GNF) (Khor et al., 2010). The Power Law Fluid is a GNF which shear stress is given by:

$$\tau = \mu \left(\frac{\partial u}{\partial y}\right)^{n} \tag{3.3}$$

Where,

η	viscosity
n	power law index

3.2.3 Viscosity Model (Cross-WLF Model)

The Cross-WLF viscosity model describes the dependency of viscosity on temperature, shear rate, and pressure. The temperature dependence is in the Willam-Landel-Ferry (WLF) form instead of the exponential form and usually more accurate to correspond to viscosity with lower temperature. The capability of this model to characterize both Newtonian and shear thinning region as well as the width range of temperature dependence resulted in the model being preferred for simulating the flow of plastic (CoreTech System, 2018). The Cross-WLF model is given by the following equation:

$$\eta = \frac{\eta_0}{1 + \left(\frac{\eta_0 \dot{\gamma}}{\tau^*}\right)^{1-n}} \tag{3.4}$$

$$\eta_0 = D_1 \, exp\left[\frac{-A_1(T - T_c)}{A_2 + (T - T_c)}\right] \tag{3.5}$$

$$T_c = D_2 + D_3 P \tag{3.6}$$

$$A_2 = \tilde{A}_2 + D_3 P \tag{3.7}$$

Where,

η	melt viscosity (Pa s)
η_0	zero shear viscosity
$\dot{\gamma}$	shear rate (1/s)
τ^*	critical stress level at the transition to shear thinning
n	power law index in the high shear rate regime
T_c	transition temperature
D_1	Newtonian viscosity at T_c

3.2.4 Volume of Fluid (VOF) Method

The volume of fluid (VOF) is a method that models the filling process by tracking the free surface of the mould (Papanikolaou & Saxena, 2021). The free surface F = F (x,y,z,t) is assumed to be a function of space and time, with a value ranging from 0 (air) to 1. (liquid). The free surface is linked to all of the simulation cells with the value (0 <F <1). The following is the VOF equation:

$$\frac{dF}{dt} = \frac{\partial F}{\partial t} + \nabla \cdot (uf) = 0 \tag{3.8}$$

In the simulation, molten PP and air were designated as two separate fluid phases. While the air escapes through the outlet vents, the molten PP fills the tray chamber during the injection moulding process. VOF monitored the progress of the flow front and the filling of the molten solder over time. The post-processing stage allows for the visualisation of the motion of molten PP.

3.3 Designing and Modelling in Solidworks

3.3.1 Design Considerations

3.3.1(a) Primary Sprue

Since the primary focus of the simulation is the intersection between the sprue, cold slug well and the runner. The dimension of the sprue is that of a standard sprue. As mentioned before, the inlet and outlet orifice are kept the same for all designs. Sprues are typically conical in shape; hence the inlet and the outlet cross-section are circular. The inlet and outlet diameters have been adapted by the design in Rusdi et. al. However there is a consideration to use the standard taper for standard sprue which is 2.386 deg. Thus, the calculation of its length is as follows:

$$B = A + L\,(\tan 2.386°) \tag{3.9}$$

Where,

A inlet orifice diameter (mm)
B outlet orifice diameter (mm),
L sprue length (mm)

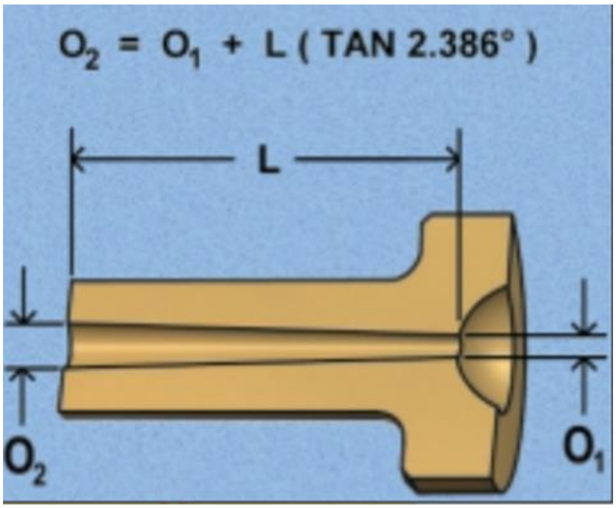


Figure 3.1 Sprue outlet orifice equation (Engelmann & Dealey, 1999)

3.3.1(b) Primary Runners

Like sprues, typical runners are usually circular since the shape has the least flow resistance. They are usually larger than the largest wall thickness of the plastic parts. This is done to avoid the holding pressure from affecting the plastic product (voids, sink marks, poorer dimensional accuracy) in case runner solidifies faster than the plastic part (Hatch, 2003).

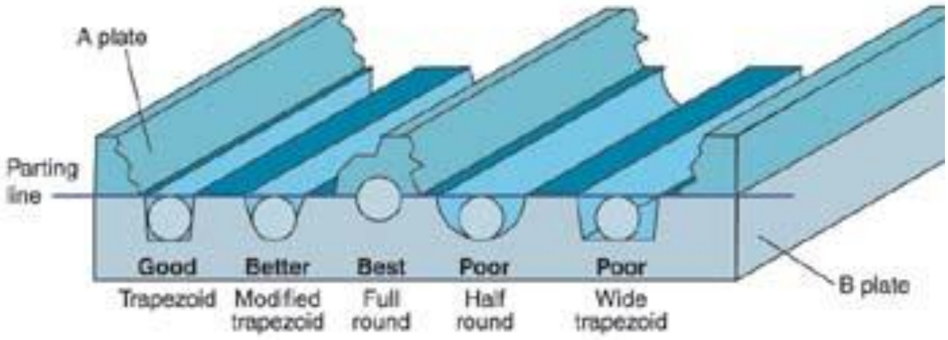
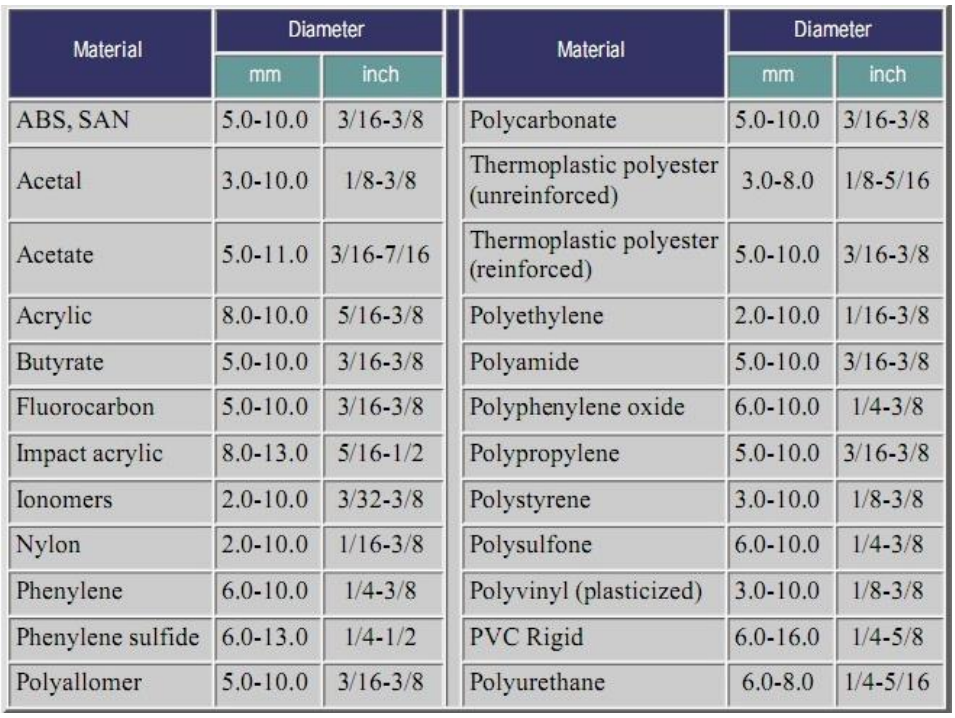


Figure 3.2 Types of runner designs (Hatch, 2003)

The minimum recommended diameter for most materials is around 1.5 mm, however the diameters are typically around 3 mm to 15 mm. In this project the smallest diameter is 2mm, the “optimum” medium diameter is 5 mm which is half the diameter of the sprue outlet orifice, and the largest size is almost the same diameter as the sprue outlet orifice.

Table 3.1 Typical runner diameters for unfilled generic materials (Zhilian Mould, 2012)



Material	Diameter		Material	Diameter	
	mm	inch		mm	inch
ABS, SAN	5.0-10.0	3/16-3/8	Polycarbonate	5.0-10.0	3/16-3/8
Acetal	3.0-10.0	1/8-3/8	Thermoplastic polyester (unreinforced)	3.0-8.0	1/8-5/16
Acetate	5.0-11.0	3/16-7/16	Thermoplastic polyester (reinforced)	5.0-10.0	3/16-3/8
Acrylic	8.0-10.0	5/16-3/8	Polyethylene	2.0-10.0	1/16-3/8
Butyrate	5.0-10.0	3/16-3/8	Polyamide	5.0-10.0	3/16-3/8
Fluorocarbon	5.0-10.0	3/16-3/8	Polyphenylene oxide	6.0-10.0	1/4-3/8
Impact acrylic	8.0-13.0	5/16-1/2	Polypropylene	5.0-10.0	3/16-3/8
Ionomers	2.0-10.0	3/32-3/8	Polystyrene	3.0-10.0	1/8-3/8
Nylon	2.0-10.0	1/16-3/8	Polysulfone	6.0-10.0	1/4-3/8
Phenylene	6.0-10.0	1/4-3/8	Polyvinyl (plasticized)	3.0-10.0	1/8-3/8
Phenylene sulfide	6.0-13.0	1/4-1/2	PVC Rigid	6.0-16.0	1/4-5/8
Polyallomer	5.0-10.0	3/16-3/8	Polyurethane	6.0-8.0	1/4-5/16

Usually, in determining the runner diameter, several things are considered such as the thickness of the runner wall, and the runner length (Zhilian Mould, 2012).

$$D = \frac{w^{\frac{1}{2}} \times L^{\frac{1}{4}}}{3.7} \tag{3.10}$$

Where,

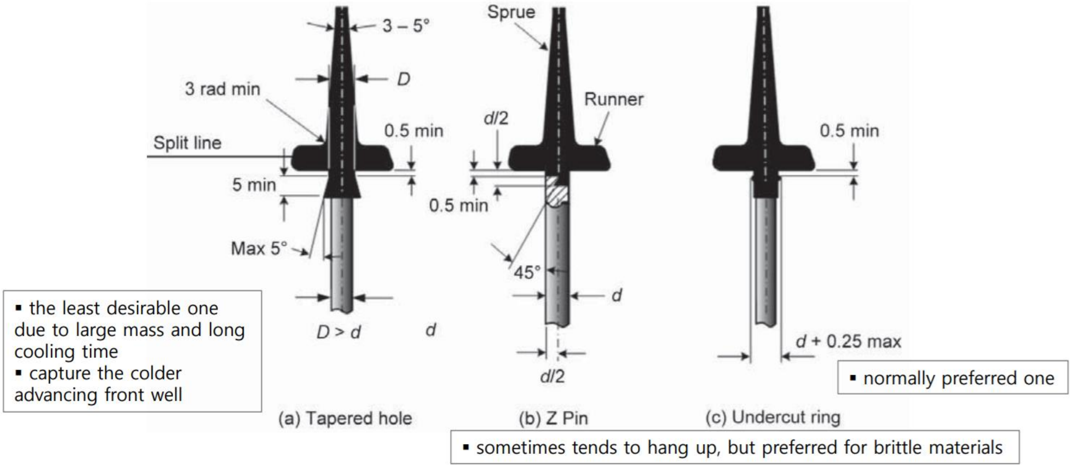
D runner Diameter (mm),
W part weight (g),
L runner length (mm)

To determine the runner length, the size of product that will be moulded, the number of gates, and the number of the cavities. Nevertheless, in this project the diameter of the runner has been set to test the recommendation where the sum of areas of the runners must be equal of smaller than the that of the sprue.

3.3.1(c) Cold Slug Well/Sprue Puller

There are many types of designs for a cold slug well, however a typical cold slug well design is an undercut ring where the well is cylindrical, has a smooth bottom, and its diameter is around the same diameter with the ejector pin. The length of the ejector pin is typically the same with the diameter of the primary runner(Engelmann & Dealey, 1999)s. The length of the well can also be larger by 1.5 -2 times than the diameter of the primary runners.

A consideration for optimising the cold slug well is that the well should not be so large that it increases the bulk of the sprue intersection as it will add to the part’s cooling time, hence its cycle time. Thus, the aim will always be to minimize the bulk size without sacrificing performance.



- the least desirable one due to large mass and long cooling time
- capture the colder advancing front well

- normally preferred one

(a) Tapered hole (b) Z Pin (c) Undercut ring

- sometimes tends to hang up, but preferred for brittle materials

Figure 3.3 Types of cold slug well designs (B.-K. Lee, 2013)

3.3.2 Sprue Design

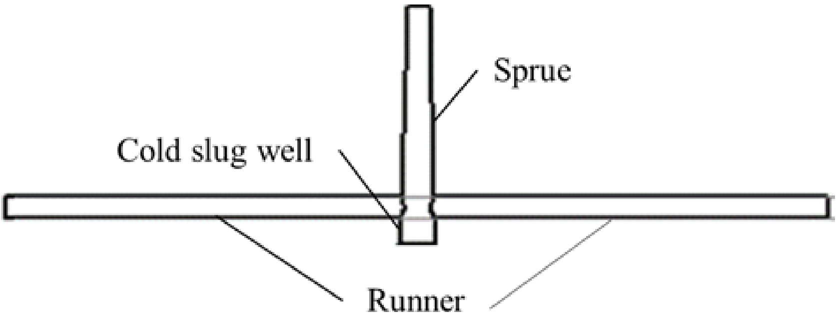


Figure 3.4 Sprue anatomy

The sprue, cold slug well and runner combination has been modelled in SolidWorks as open ended without gates and cavities at the end of its primary runner. This design has been made with reference to the dimensions made in the previous section. The following is the full dimension list of the sprue which partially based on the design by Rusdi et al:

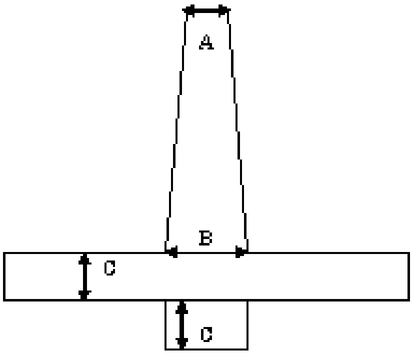


Figure 3.5 Sprue dimensions (Polyplastics, 2012)

Table 3.2 Sprue dimensions

Dimension	**Symbol**	**Value**
Taper	θ	1.73°
Inlet Orifice Diameter	A	6 mm
Outlet Orifice Diameter	B	10 mm
Runner Diameter	C	5 mm
Well Length	D	5 mm
Length	L	65.5mm

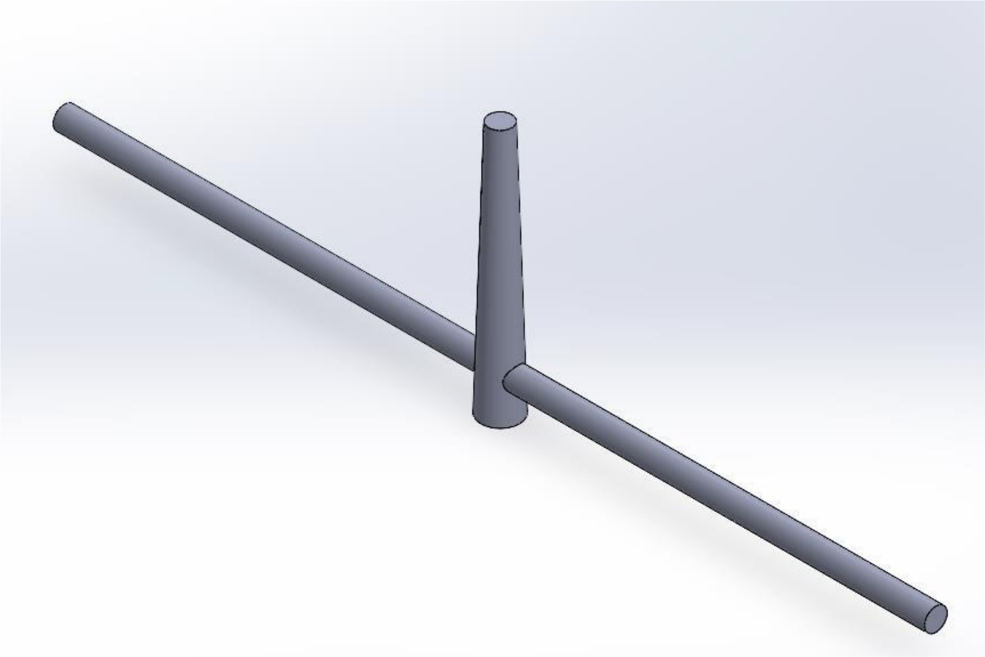


Figure 3.6 3D model of sprue

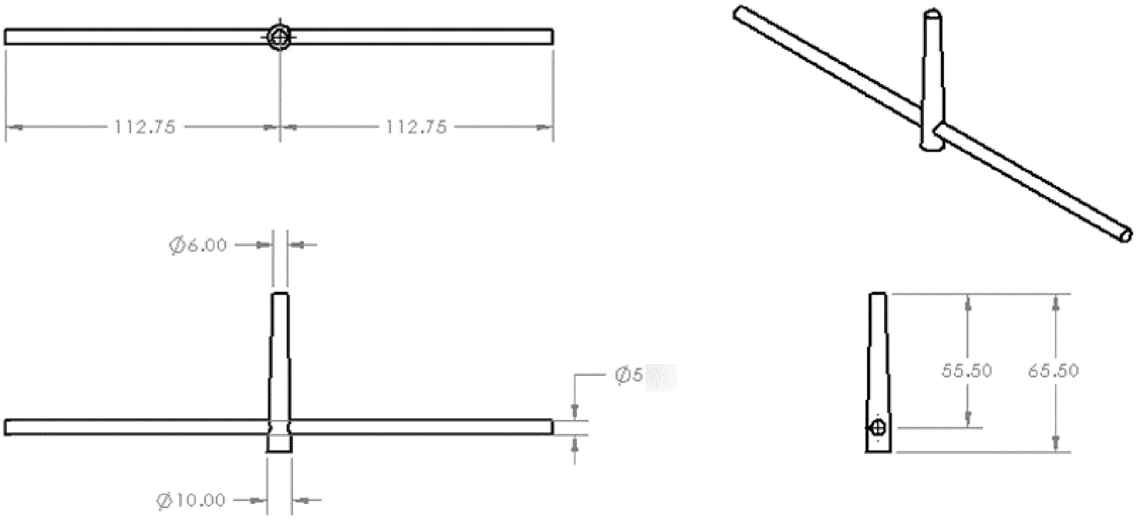


Figure 3.7 Sprue dimensions

3.4 Simulation Setup in ANSYS Fluent

The following is the flowchart which depicts the overall simulation process in ANSYS Fluent:

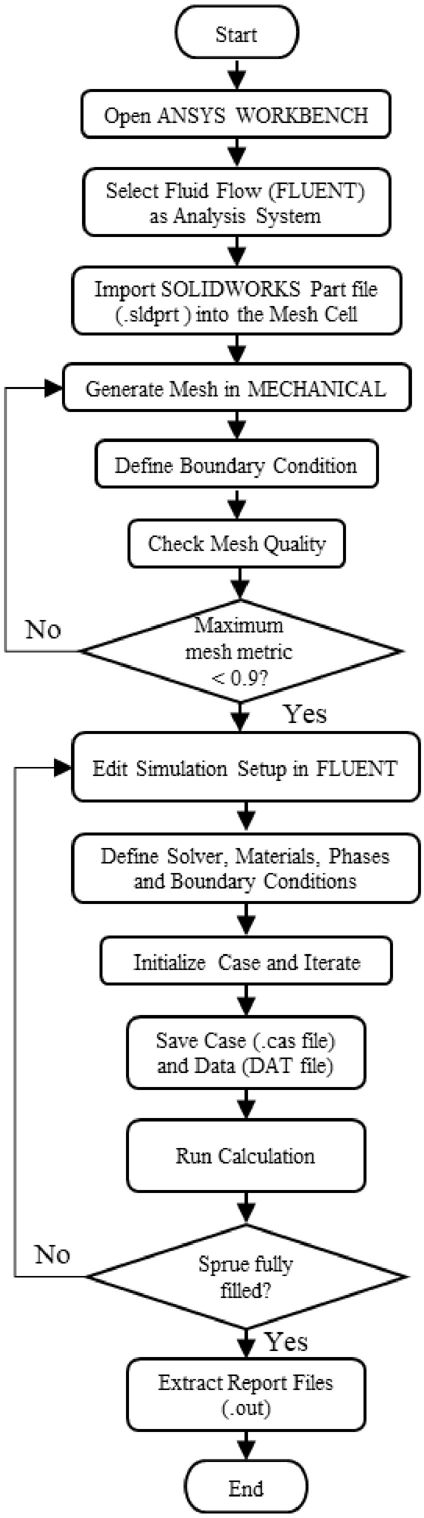


Figure 3.8 Simulation process flowchart

3.4.1 Mesh development in ANSYS Mechanical

For the ANSYS Simulation, the part has been exported and meshed in ANSYS Mechanical through ANSYS Workbench. The size of the mesh is 2 mm, which results is the total element of 66663. The boundaries of the sprue have been set to identify the inlet, oulet, and walls of the sprue. On the other hand, the setup and parameters that will be used in ANSYS Fluent is done in reference to the parameters in the research paper by Rusdi et. al.

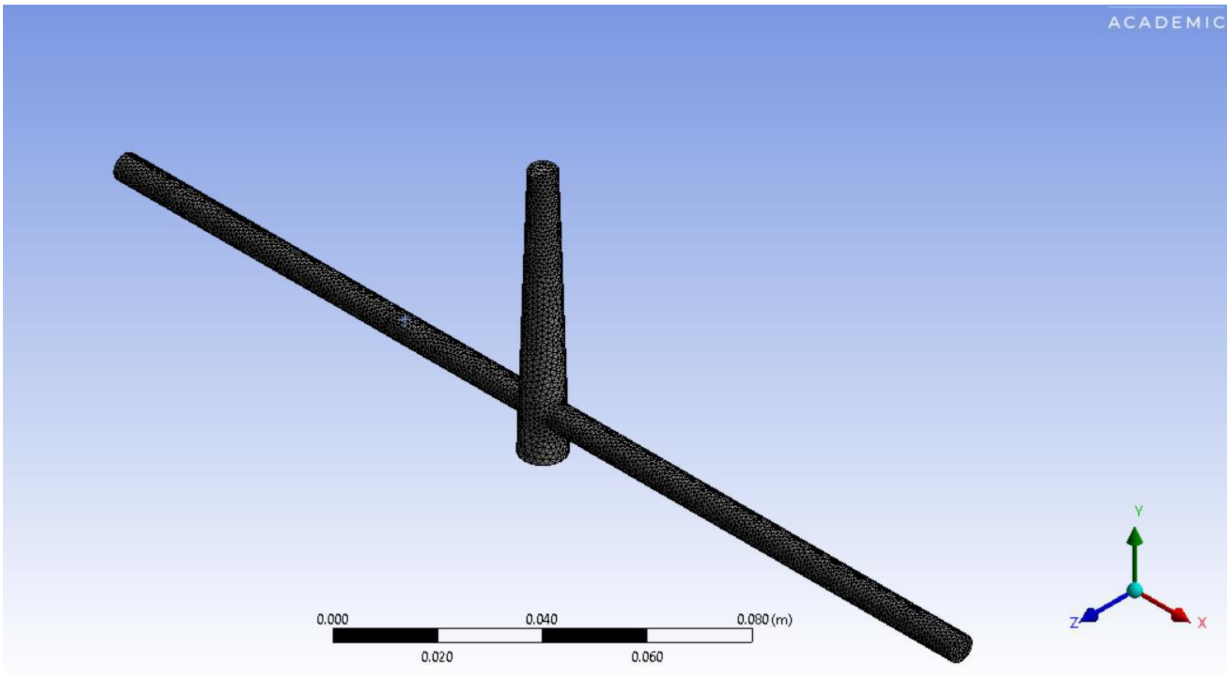


Figure 3.9 Meshing of the sprue

3.4.2 Boundary Condition Definition

The model's boundary conditions are then established in the computational domain. The boundaries are comprised of the inlet, the outlets, and the gate (combination of sprue, cold slug well and runner) wall. The inlet is orifice of the primary sprue at the end of the sprue, where the molten plastic enters the gate. On the other hand, the outlets are the orifices of the runners at the end of each runner, where the molten plastic exits the gate. The boundaries can be referred with the following figures:

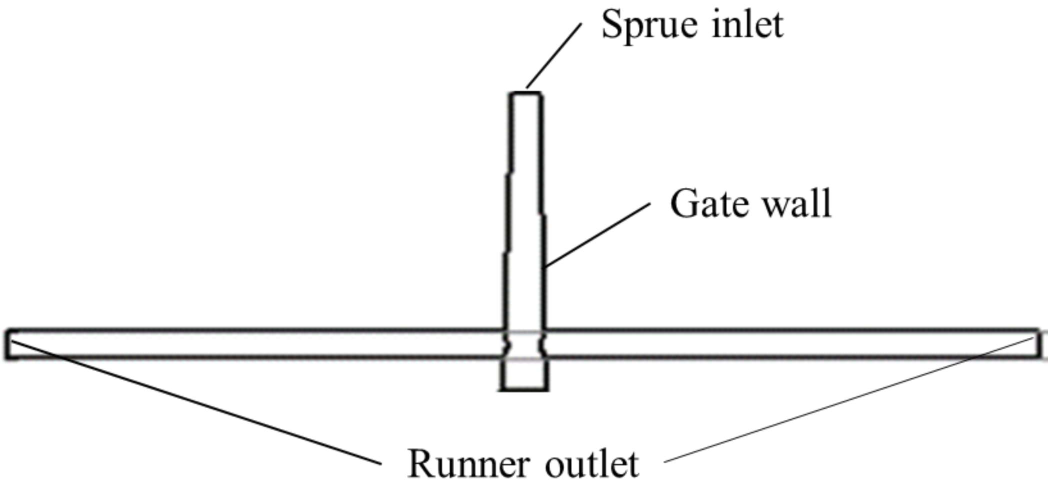


Figure 3.10 The gate system boundaries

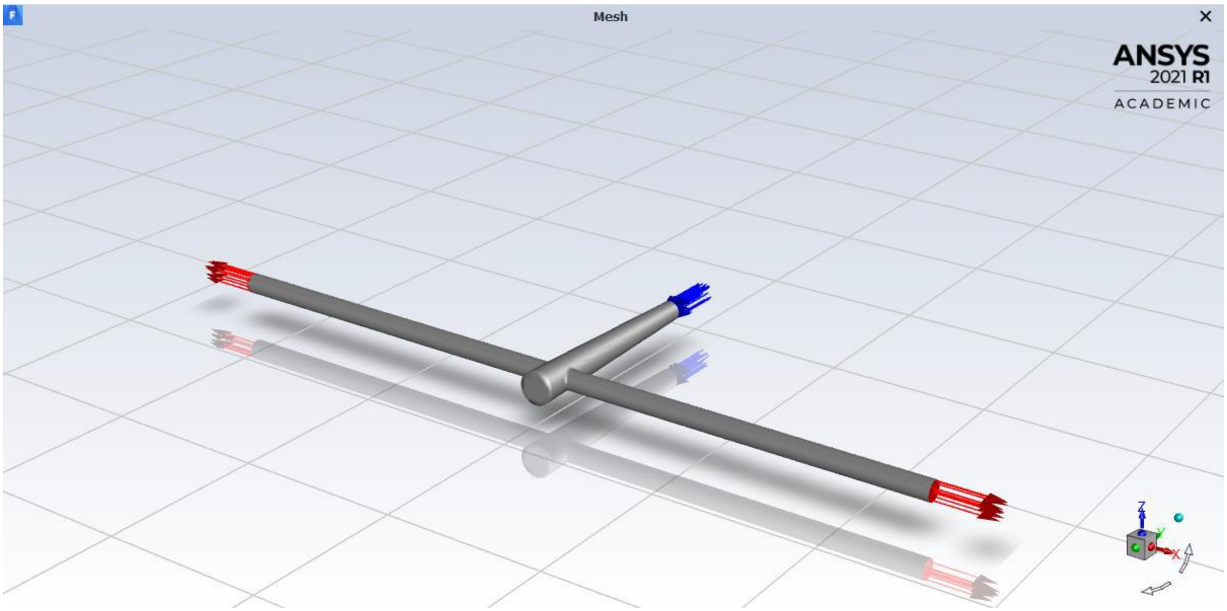


Figure 3.11 Boundaries of the sprue

The boundary and initial conditions can be described with the equations below(Rusdi et al., 2016):

Table 3.3 Boundary and Initial Conditions

Part	**Boundary/Initial Conditions**
Melt Front	$p = 0$ (3.11)
Gate Wall (Sprue, well, runner)	$u = v = w = 0$ $T = T_w$ (3.12)
Sprue Inlet	$p = p_{in}(x, y, z)$ $T = T_{in}$ (3.13)

3.5 Material, Solver and Phases

For the wall boundary, the solid is set as tool steel. For the fluid, two types of polypropylenes (PP) are used in this study which are medical grade PP (LB6331) and commercial grade PP (TP340). The study is conducted for temperatures 180°C, 190°C, 200°C, 210°C, and 220°C for LB6331 whereas for 190°C for TP340. Thus, the mechanical properties of the PP are set at 200°C, while viscosity is set as a cross model according to the inlet pressure. The rheology data that will be used for the cross model of both grades are obtained from a study by Rusdi et. al. which are obtained from the GÖTTFERT Rheograph 25 through the WinRheo II software as follows:

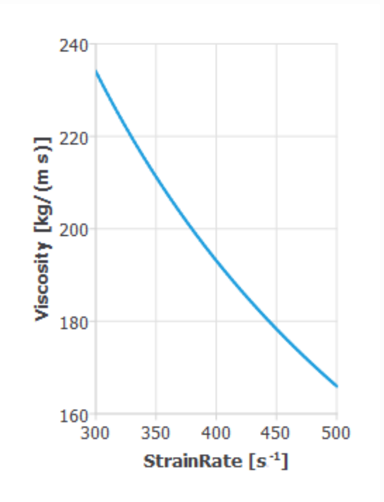
Table 3.4 LB6331 rheology data

T [°C]	**180**	**190**	**200**	**210**	**220**
η_0 [Pa s]	2.67E+03	2.16E+03	1.77E+03	1.52E+03	1.37E+03
η [-]	2.39E-01	2.59E-01	2.81E-01	3.05E-01	3.12E-01
λ [s]	6.30E-02	5.73E-02	5.26E-02	4.90E-02	4.61E-02

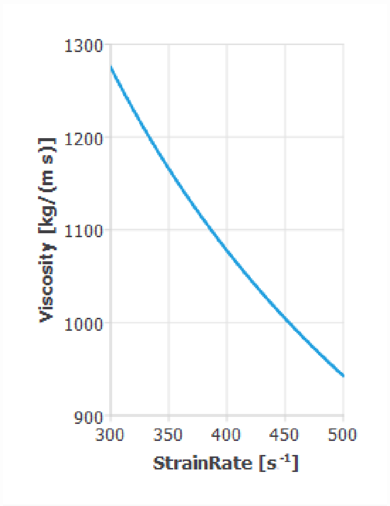
Table 3.5 TP340 rheology data

T [°C]	**190°C**
η_0 [Pa s]	7.83E+03
η [-]	3.12E-01
λ [s]	3.60E-02

The following are the graphical representation of the cross model for PP at 190°C for PP grade LB6331 and TP340 used in the simulation.



Graph 3.1 LB6331 Polypropylene Cross Model



Graph 3.2 TP340 Polypropylene Cross Model

3.6 Data Collection

The data collected are the runner pressure at the inlet, middle and outlet, overall velocity and volume fraction at the runner. The raw data are extracted from the .out file written by FLUENT. The data are then converted in excel, tabulated, and analysed.

3.7 Simulation Accuracy

In numerical analysis, two of the things that can impact a numerical simulation are the time step size and the grid resolution range. It is important to conduct a time step study as well as a grid dependency test to reduce any possible biases that can affect the accuracy of the numerical analysis whereby the results is either overestimated, or underestimated (M. Lee et al., 2020; Rusdi et al., 2016). These tests are also conducted to optimise the accuracy with respect to the computation time and resources limitation to reach an optimal balance between time and effort (Hockley, 2021).

3.7.1 Time Step Study

To conduct the time step study, five time steps sizes are selected for the study, i.e. 5×10-2, 5×10-3, 5×10-4, 5×10-5 and 5×10-6. The test is conducted by simulating the gate design in Figure 3.6 with 66663 tetrahedral mesh elements, at 200°C melt temperature, 47MPa of system pressure and the fluid properties of LB6331. The parameters of the injection moulding process are kept constant for all time step sizes. The time steps are compared according to the resultant runner pressure.

Table 3.6 Time step study results

	Pressure, P (Pa)		
Time Step Exponent, 5e-n	**Inlet**	**Middle**	**Outlet**
2	9896.943	334.3697	0.3064384
3	14599.77	5203.884	0.689762
4	10047370	-658.6726	-0.3987542
5	31428810	15991490	1078.892
6	31428810	15991490	1078.892