Design of Surface Textures for Improved Resin Replenishment During Additive Manufacturing

BY

Tiantian Ma
B.S., Polytechnic University of Turin, Italy, 2014

THESIS
Submitted as partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering in the Graduate College of the University of Illinois at Chicago, 2016

Chicago, Illinois

Defense Committee:

Jie Xu, Chair and Advisor
David Eddington, Bioengineering
Pietro Asinari, Politecnico Di Torino
ACKNOWLEDGEMENTS

This thesis would not have been possible unless professor Gola introduced me to the exchange program which I admired, instructed me to prepare both knowledge and documents, and helped me to land in UIC and do the project I liked so much.

I owe my deepest gratitude to my thesis advisor Professor Jie Xu who helped me whenever I went to him, guided me back to the right direction when I headed a wrong way, answered so patiently to tons of questions, corrected my numerous mistakes, and supported me when I was frustrated.

I am grateful to my co-professor Asinari, who supported me a lot in a numerous ways although he was in Italy and concerned with not only every tiny detail of my project and followed the updates, but my personal life and the situation I got. He was really professional and full of responsibility.

I would like to thank my thesis committee member Professor David Eddington for helping me improve my thesis as well as Yayue Pan who has the latest and most advanced knowledge about the topic of my project and was generous in sharing her ideas and knowledge with me

I would also like to thank my colleague, PhD student Dmitry, who had a perfect understanding of my project and cared about every change and update. Furthermore, he mastered plenty of knowledge about physics which helped to explain plenty of my misunderstandings.

It is an honor to thank those who made this thesis better.
PREFACE

It was the summer of 2014, I was in Polytechnic University of Torino, Italy, just finishing my last semester of bachelor. Professor Muzio Gola, who is full of energy and intelligence, offered me a challenging program: to apply for double degree program in Polito and UIC. I was so nervous and excited, because I’ve desired to U.S. for several years but I had too many friends in Italy and I really loved the way Italians live. It took me one week to make a decision on acceptance of the offer.

However to match the requirements of the program, I had to work really hard. I spent busy and solid two semesters, and I had strong interest in the course ‘Advanced engineering thermodynamics’ with Professor Asinari, which was about the fluid dynamics and thermo dynamics.

Back to the morning of March 27th 2015, Professor Asinari informed me that it’s time to choose professors. After I checked every professor on the list, I went to Professor Xu without hesitation as his research area was fluid dynamics which was my favorite one. Fortunately, Professor Xu accepted me and offered me some topics.

Things didn’t go smoothly after I initially chose a project to work on, because I spent a lot of time gaining basic knowledge on my topic of interest. This strategy resulted in stuck with classes for my second semester. Any mistake would lead to failure of matching the requirements of the programs, and if that happened every effort I did couldn’t pay off. Therefore, I started to focus on my classes again and fortunately I matched up after solving myriad of problems I faced.
PREFACE (continued)

I still remember 13th August 2015. It was my big day, I would take the flight from Milan to Chicago. I was so excited about landing in the U.S. but I still loved Italy. However, at the very beginning it was really tough. I didn’t have a place to live and I didn’t have a license, the worst thing was that I knew nobody except but Lynn Thomas and Professor Xu.

I was and I am still so grateful to Lynn, who helped me out, who taught me to register and everything. With her patient help my life in the U.S. was settled down and I could completely focus on my classes and thesis. However, problems still came out during the studies. The process in US was totally different compared to the process in Italy.

At this period, Professor Xu instructed me and supported me a lot, helped me get through all my difficulties and misunderstandings. When I finished my first semester in the U.S. with top grades for all the classes, I was very proud of myself. My hard work paid off and I became more confident that I could handle my thesis and finish it well. But things didn’t go in the direction I thought.

My thesis is about the improvement of the performance of a 3D printer through numerical model analysis. The good thing was that it was about fluid dynamics which I was strongly interested in. The bad thing was that I had to learn a totally new software program to help me do the iteration and get the solutions. I had a basic understanding of ANSYS Fluent to perform the simulation and get the solutions. But when I tried to start, I didn’t know where to start.

I had many discussions with Professor Xu, who helped me gain a deep understanding of the entire project and possessed profound knowledge about it.
What was more, I watched dozens of videos in YouTube trying to gain as much experience as I could through all the possible means.

Little by little, step by step, I finished my model first and then I attended the training covering the meshing process. During the training, I finished my first meshed file, which made me proud even thought it was not perfect. Since I became more and more familiar with both my project and ANSYS Fluent, I was confident that I could solve the problem.

This experience will be treasured by me forever, it’s a period mixed with sweat, tears and laughs, it’s a trip that I went through sorrows, frustrations and successes, it’s a memory about working hard with my Professor and my friends, and playing hard with them.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>CHAPTER</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1 3D Printing definition</td>
<td>1</td>
</tr>
<tr>
<td>1.2 3D printing processes</td>
<td>4</td>
</tr>
<tr>
<td>1.3 Project background</td>
<td>9</td>
</tr>
<tr>
<td>NUMERICAL METHODS</td>
<td>12</td>
</tr>
<tr>
<td>2.1 Computational fluid dynamics</td>
<td>12</td>
</tr>
<tr>
<td>2.1.1 CFD introduction</td>
<td>12</td>
</tr>
<tr>
<td>2.1.2 CFD simulation process</td>
<td>13</td>
</tr>
<tr>
<td>2.2 Finite methods</td>
<td>16</td>
</tr>
<tr>
<td>2.2.1 Finite element method</td>
<td>16</td>
</tr>
<tr>
<td>2.2.2 Finite volume method</td>
<td>17</td>
</tr>
<tr>
<td>2.2.3 Finite difference method</td>
<td>17</td>
</tr>
<tr>
<td>2.2.4 FVM Vs FEM</td>
<td>19</td>
</tr>
<tr>
<td>2.3 ANSYS Fluent</td>
<td>20</td>
</tr>
<tr>
<td>2.3.1 Procedure</td>
<td>20</td>
</tr>
<tr>
<td>2.3.2 Geometry</td>
<td>21</td>
</tr>
<tr>
<td>2.3.3 Mesh</td>
<td>23</td>
</tr>
<tr>
<td>2.3.4 Set up the Simulation in ANSYS Fluent</td>
<td>28</td>
</tr>
<tr>
<td>2.3.5 Check mesh</td>
<td>29</td>
</tr>
<tr>
<td>2.3.6 Solver</td>
<td>34</td>
</tr>
<tr>
<td>2.3.7 Post-Processing</td>
<td>40</td>
</tr>
<tr>
<td>RESULTS</td>
<td>48</td>
</tr>
<tr>
<td>3.1 Model validation</td>
<td>48</td>
</tr>
<tr>
<td>3.2 Pressure analysis</td>
<td>48</td>
</tr>
<tr>
<td>3.3 Velocity character</td>
<td>53</td>
</tr>
<tr>
<td>CONCLUSION</td>
<td>58</td>
</tr>
<tr>
<td>CITED LITERATURE</td>
<td>60</td>
</tr>
<tr>
<td>VITA</td>
<td>69</td>
</tr>
</tbody>
</table>
## LIST OF TABLES

<table>
<thead>
<tr>
<th>TABLE</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>EXAMPLES OF NET SHAPE AND SUBTRACTIVE PROCESSES</td>
</tr>
<tr>
<td>II</td>
<td>COMPARISONS BETWEEN FINITE METHODS</td>
</tr>
<tr>
<td>III</td>
<td>ADVANTAGES AND DISADVANTAGES OF CUT CELL MESHING</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>FIGURES</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Conventional 3D-printed sample</td>
</tr>
<tr>
<td>2</td>
<td>Brief history of additive manufacturing process</td>
</tr>
<tr>
<td>3</td>
<td>Traditional process</td>
</tr>
<tr>
<td>4</td>
<td>Clip idea</td>
</tr>
<tr>
<td>5</td>
<td>Clip process</td>
</tr>
<tr>
<td>6</td>
<td>CFD procedure</td>
</tr>
<tr>
<td>7</td>
<td>CFD simulation</td>
</tr>
<tr>
<td>8</td>
<td>Microgrooves pattern design</td>
</tr>
<tr>
<td>9</td>
<td>Illustration of textured window top view</td>
</tr>
<tr>
<td>10</td>
<td>Top view and side view of flat pattern</td>
</tr>
<tr>
<td>11</td>
<td>Schematic of separation forces for solid circular geometry</td>
</tr>
<tr>
<td>12</td>
<td>Part elevation height</td>
</tr>
<tr>
<td>13</td>
<td>Part elevation velocity</td>
</tr>
<tr>
<td>14</td>
<td>Flat pattern simulation compared to theory results</td>
</tr>
<tr>
<td>15</td>
<td>Force comparison texture pattern with theoretical calculation</td>
</tr>
<tr>
<td>16</td>
<td>Mesh 8 microgrooves pattern</td>
</tr>
<tr>
<td>17</td>
<td>Mesh with cross microgrooves pattern</td>
</tr>
<tr>
<td>18</td>
<td>Inflation outside boundary</td>
</tr>
<tr>
<td>19</td>
<td>Dead zone</td>
</tr>
<tr>
<td>20</td>
<td>Mesh independent study</td>
</tr>
</tbody>
</table>
21 Type of mesh cells .......................................................................................... 34
22 Different types of 3D mesh cells .................................................................. 35
23 Single-block geometry ................................................................................... 35
24 Logical representation ................................................................................... 36
25 Mesh with 8 microgrooves pattern ................................................................. 37
26 Case of mesh size warning ............................................................................. 38
27 Changes in chemical properties Photopolymer at Wikipedia ...................... 40
28 Formation of cavities ..................................................................................... 48
29 Force of 4 different textured surface ............................................................... 49
30 Example liquid flow under vacuum pressure .................................................. 51
31 Pressure scale ................................................................................................ 51
32 Flat surface pressure distribution at time 0.6s (left) and 1.0s (right) ............... 52
33 Cross pattern pressure distribution at time 0.6s (Left) and 1.0s (Right) ........... 52
34 12 Radial pattern pressure distribution at time 0.6s (Left) and 1.0s (Right) ....... 53
35 8 radial pressure distribution at time 0.6s (Left) and 1.0s (Right) .................... 54
36 Velocity scale .................................................................................................. 55
37 Velocity distribution of 12 microgrooves pattern ............................................ 56
38 Contours of 8 radial microgroove pattern velocity ........................................ 56
39 Cross section microgrooves pattern ............................................................... 57
## LIST OF EQUATIONS

<table>
<thead>
<tr>
<th>EQUATION</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 FEM governing equation</td>
<td>15</td>
</tr>
<tr>
<td>2 FVM governing equation</td>
<td>16</td>
</tr>
<tr>
<td>3 FDM governing equation</td>
<td>17</td>
</tr>
</tbody>
</table>
# LIST OF ABBREVIATIONS

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AM</td>
<td>Addictive manufacturing</td>
</tr>
<tr>
<td>FDA</td>
<td>Food and drug administration</td>
</tr>
<tr>
<td>CLIP</td>
<td>Continuous liquid interface production</td>
</tr>
<tr>
<td>PDMS</td>
<td>Polydimethylsiloxane</td>
</tr>
<tr>
<td>SLA</td>
<td>Stereolithography</td>
</tr>
<tr>
<td>SLS</td>
<td>Selective laser sintering</td>
</tr>
<tr>
<td>FDM</td>
<td>Fused deposition modeling</td>
</tr>
<tr>
<td>ASTM</td>
<td>American Society for Testing and Materials</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational fluid dynamics</td>
</tr>
<tr>
<td>CV</td>
<td>Control volume</td>
</tr>
<tr>
<td>IC engine</td>
<td>Internal combustion engine</td>
</tr>
<tr>
<td>FVM</td>
<td>Finite volume method</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite element method</td>
</tr>
<tr>
<td>FDM</td>
<td>Finite difference method</td>
</tr>
<tr>
<td>N-S equation</td>
<td>Navier–Stokes equations</td>
</tr>
<tr>
<td>UV</td>
<td>Ultraviolet</td>
</tr>
<tr>
<td>BC</td>
<td>Boundary Condition</td>
</tr>
<tr>
<td>IT</td>
<td>Information technology</td>
</tr>
<tr>
<td>UA</td>
<td></td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>------------------------------</td>
</tr>
<tr>
<td>PLA</td>
<td>Poly lactic acid</td>
</tr>
<tr>
<td>CFX</td>
<td>Computational fluid professional</td>
</tr>
<tr>
<td>HEX</td>
<td>Hexahedron</td>
</tr>
<tr>
<td>TET</td>
<td>Tetrahedral</td>
</tr>
<tr>
<td>TRI</td>
<td>Trihedral</td>
</tr>
</tbody>
</table>
SUMMARY

3D printing technology has been growing rapidly. Since 1987 when the first additive manufacturing machine was invented, researchers were mainly focused on solving two problems: achieving better printing quality and increasing process speed. 3D printing differs from traditional manufacturing in a way it creates the object, namely by accumulating material instead of removal.

Current 3D printing process is done using layer by layer fashion. In contrast, a new way of production, continuous liquid interface production (CLIP), is layerless and based on vat polymerization. It provides nearly 100 times faster building speeds compared to traditional 3D printing.

One question comes with this high efficient 3D printing process is that resin replenishment is not fast enough for parts with large cross sections. As the built part keeps going upward, pressure gradient is created that drives a resin flow into the gap between the oxygen permeation window and the bottom of the build part. The gap size is dictated by the thickness of the oxygen diffusion be very thin. Therefore, the separation force and suction pressure are very high, that appears to which is the main problem for this new layerless printing process. Resin flow speed can be changed with different microgroove patterns. With a large cross section, the replenishment of polymer cannot reach the limit before polymerization happens. Resin flow rate is a great limit in 3D printing manufacturing object with wide cross section.

To eliminate the limitation, the resin flow should be speed up. This project designs models with different microgroove patterns to address this issue.
SUMMARY (continued)

In ANSYS Fluent different kinds of microgroove patterns are simulated and the results are compared for the identification of optimal geometry. Three possible microgrooves are designed by experience. With the same speed of elevation, different flow velocity and pressure are compared. The lowest pressure point and the total separation force are quantified. The results in ANSYS Fluent will help us find the best design of window microgroove pattern and make 3D printer more efficient for fabricating objects with wide cross section areas.
CHAPTER 1

INTRODUCTION

1.1 3D Printing Definition

Nowadays 3D printers are becoming more and more common [1] since they are fast, efficient and can do personalized manufacturing [2]. It is simple to understand 3D printing as an additive manufacturing (AM) process which makes 3D solid objects from a virtual design by accumulating materials [3]. 3D printed objects are made by using accumulating processes in which an object is created by combining sliced layers [4] and each slice is a cross-section of the product which is thin and horizontal [5]. It starts with a virtual model in CAD with 3D-modeling programs or a 3D scanner [6]. 3D printing makes producing any kind of geometry model possible and easy [7].

Figure 1.Conventional 3D-printed sample
3D printing has reached relatively high speed and resolution [1]. In Figure 1, a red rabbit was made by a commercial 3D printer. The object can also be fabricated by milling and other methods, but 3D printing is the best way to produce complex models like this quickly and easily [8].

It is almost impossible to ignore the widespread application of 3D printing which is an unstoppable trend when looking back into the history of manufacturing [9]. The process presents a gradual transition from man-made to machining [10]: the more advanced the technique is, the less human activity is involved during manufacturing processes. There are several classes of manufacturing process [11]: net shape process, subtractive process, and additive process. Net shape and subtractive processes are summarized in TABLE I:

<table>
<thead>
<tr>
<th>Net shape process</th>
<th>Subtractive process</th>
</tr>
</thead>
<tbody>
<tr>
<td>-Forging, drawing, extrusion, rolling</td>
<td>-Lathing, milling grinding, drilling</td>
</tr>
<tr>
<td>-Sheet metal forming, bending</td>
<td>-Water jetting, laser cutting, etc.</td>
</tr>
<tr>
<td>-Die casting, investment casting</td>
<td></td>
</tr>
<tr>
<td>-Injection modeling</td>
<td></td>
</tr>
</tbody>
</table>

AM is a class of manufacturing technology including 3D printing [12], which is a method to create objects by adding material to the object slice by slice accurately [13]. It has taken a long time for 3D printing’s evolution.

A brief history of AM is shown in Figure 2. Since the first AM system made in 1987, it developed very fast in terms of reducing price and increasing speed.
Actually, at first AM was used on the tool room of spectrum manufacturing [14]. For example, rapid prototyping was one of the earliest approaches, aiming to decrease both the working time and investment for new generation machine[15], which earlier can only be done with the subtractive process and that was more expensive and slower [16]. However, as time went by and with the development of the technology, additive methods started to play more and more important roles in manufacturing [17]. Products that were formerly done only by subtractive methods can now be made more profitably via additive methods.
There are several outstanding advantages of addictive methods that cannot be ignored. First, it is possible to change the design models easily and quickly [18]. Whether they are complicated or not, no more additional tools are required besides CAD. 3D printing also reduces production time, shortens supply chains and makes lightweight structures. Therefore it is regarded as a green manufacturing technology [19]. Moreover, 3D printing opens up opportunities for innovation, as it is capable of making more complex parts as well as multi-functional parts [20].

1.2 3D Printing Processes

3D printing starts with virtual models based on the objects that need to be created [21]. Virtual models are built in CAD files or scanned by 3D scanners. A 3D scanner builds a 3D digital copy of an object by using different technologies.

Recently, researchers are investing on hardware to have better performance of 3D scanning [22]. 3D scanners can also be used in small devices. For example, in the future an integrated 3D scanner will be used in smartphones [23]. Digitizing real objects into 3D models will become as easy as taking a picture.

3D scanning with a professional HDI 3D scanner uses structured light to prepare a digital file for printing [24]. As long as the model is ready in 3D scanning, the next step would be to build via a 3D printer. The objects were sliced into thousands of pieces. The 3D printing works slice by slice. Every slice works like a 2D printer. By repeating again and again the 2D printing, the 3D model will be shown.

3D printers use different technologies [25]. There are several methods, which are different
from each other in the way that the final object is made. Melting or softening materials are most used to build the layers. Another way is using a curing photo-reactive resin with a UA laser [26] or another similar power source to fabricate one slice at a time. Based on this technology the most common one is called stereolithographic (SLA).

Traditional 3D printing uses slice-by-slice printing method. Final production is delaminated into thousands of pieces. After the object absorbs enough light energy for one slice [27], the elevation goes upward, and resin will be recoated. The part elevation goes down. After reposition the new working cycle will start as shown in Figure 3.

![Traditional Process: Layer-by-Layer Printing](image)

*Figure 3. Traditional process*
1. **Vat Photo polymerization**

In a vat photo polymerization process [28], UV light is supplied to harden the resin when needed. Plastics and polymers are the main materials for vat polymerization. Supports are needed, after all the processes the parts are removed by a knife. It is relatively expensive, but the accuracy is relatively high [29], and entire process is highly efficient.

2. **Material extrusion** [28]

Fuse deposition modeling (FDM) [29] is widely used since it is relatively inexpensive. The materials are in a melted state, and layers are easily fused together. The final product quality is largely dependent on the radius of the nozzle. During the process, constant and stable pressure is needed [9], which in some cases is a limitation.

3. **Material jetting** [30]

Multiple nozzles are used [31]. The part is cured under the UV light and produces smoothly finished object. This is the only process that can combine different print materials within the same job. Support material is needed. At the end of the process, the support material can be removed in a high powered water jet station [32].

4. **Binder Jetting**

The build material and binder are both used at the same time. The binder jetting method is referred to as a 3DP printer. The Binder works like adhesive powder [33] and the layers are adhered by the binder. The material used in binder jetting is in powder form. The powdered
material [34] can be metal, polymer or ceramic. One of the advantages of this technology is that no support is needed during the process because it is self-supported. The overall process time is significantly enlarged since the binder material needs time to obtain mechanical and structural stability [35].

5. Powder bed fusion

The material is in powder form. The common powder is typically metal or polymers. A laser shapes the powder into the form required. The new powder stock is then added forming a new layer, thus building in a slice by slice fashion.

There are several disadvantages of this method. For example, the finished size has limitations, and the working speed is relatively low [36]. But the machine is relatively inexpensive.

6. Sheet lamination [37]

A laser or knife will cut the materials (paper, plastic, sheet metals) into the required shape. Then the next slice will be added using sheet lamination method.

7. Directed energy deposition [38]

The directed energy deposition method is also called the electron beam melting process. Material is in the form of wire or powder. Only metal is used not polymer or ceramic. This process can be used to repair or add materials to the required parts. However, high speed leads to low efficiency [39], and vice versa.
8. Continuous liquid interface production (CLIP) [40]

CLIP is a new revolution in 3D printing history. The most attractive features are the increased resolution and speed. Traditional way is slow and with low resolution. CLIP increased the speed 100 times compare to the traditional 3D printers. Continuous Liquid Interface Production is a new way in additive manufacturing production famous for sliceless 3D printing [41]. The materials used are photo polymers. But continuous liquid interface production can create a smoother and wider variety of shapes.

![Figure 4. Clip idea](image_url)
In traditional 3D manufacturing, time is wasted between finishing prior slice finished and when the new slice starts. The motion of the finished part moving up and down challenges the support structure.

As with the traditional way, a projector to supply the light energy is also needed in this process. The UV light goes through the oxygen window and reaches the bottom of the build[42]. The material used here is photo polymer. Platforms are the same with the traditional method.

The main difference is that the new way has an oxygen containing dead zone[43]. The dead zone works as a barrier between the solid part and liquid.

The dead zone stopped the unwanted liquid solidification. Photopolymer resin will not attach to the window. This is the key point to keep printing process continuous.

This process overcomes the critical problem in 3D printing. The production time can be hugely shortened from hours or days to minutes. Besides time saving, the production resolution can be higher than 100µm [44].

![Figure 5..Clip process](Image)
1.3 Project Background

3D printers have a lot of advantages such as cheap manufacturing, quick prototype production with complex structures [45], and less waste. However, there are still limitations of 3D printers like low speed, low resolution, and high price. Currently, depending on the size of the model and quality of the printer, it takes hours to days for printing an object. Understandably, these metrics are all related, and one has to make tradeoffs among them. For example, there is a direct relation between price and processing time. Resolution is determined by the material used and machine structure and so on[46]. It also affects processing time significantly. 3D printers with faster building speed and shorter post processing time at a higher resolution are highly desirable.

Continuous liquid interface production (CLIP) has achieved a very high speed and resolution level. This process does not need to stop the motion of part elevation after each slice is completed. Time spent for waiting for resin to recoat can be saved. The requirement for resin flow rate is also very high. In CLIP, resin flow rate has been highly improved to 600mm/h however it is still very weak in fabricating objects with large cross sections.

As shown in Figure 4, the part elevation is continuously going up, and suction pressure gradient drives resin flow. Resin flow rate is influenced by many factors such as part geometry, viscosity of resin and surface texture. Current CLIP technology only controls part elevation speed and cannot control the other factors like part geometry, window condition and resin curing.

Surface texturing has been used to alter pressure gradient in this study. Two kinds of geometry are designed: radial microgroove pattern and cross microgroove pattern. In both models the sum
of volume in each groove pattern is the same. Surface texturing is performed on a 100μm thick Teflon film and a 1mm thick PDMS film [47].

The so called separation force is the limit force to ensure resin attachment with the part continuously to elevate. This can be obtained both in numerical or experimental methods. Calculations are made with the assumption that the resin properties is stable under the specific temperature and that the textured surface is rigid. The pressure in the center can be calculated by N-S momentum equation and mass conservation equations.

A good surface texturing design can speed up resin flow and resin replenishment. It is a perfect solution for 3D printing that deals with large cross section objects.
CHAPTER 2

NUMERICAL METHODS

2.1 Computational Fluid Dynamics

Computational fluid dynamics is one of the important branches of study in fluid dynamics. It uses numerical methods and algorithms to solve the problems in fluid. Since fluid flow is complex and the calculation needs iterations, computer performance is essential in computation.

2.1.1 CFD Introduction

According to fluid dynamics, the dynamics of fluids like gases and liquids is governed by the partial differential equations. They follow the general laws of conservation of mass [66], momentum and energy. What CFD does is the discretization continuous functions to infinite intervals, in which conservation equations are applied and these equations can be solved by numerical methods [66]. Obviously calculations done by computers are accurate and efficient [68], and that is the most outstanding advantage of CFD. However, because information is lost during discrete process, CFD simulation can never represent 100% realities.

The so-called numerical modeling refers the discrete process through meshing and grids. With numerical models, solvers like Fluent can do iterations and get solutions. Figure 6 shows the procedures of a complete engineering design with the help of CFD. When the numerical results do not match the theoretical predictions, the design needs modification.
CFD assists in predicting fluids performance. In general, much more time and money is needed to do plenty of experiments in order to do what CFD can easily do. What’s more, it’s impossible to control the experimental conditions to be the same every time. Specific procedures will be introduced below.
2.1.2 CFD Simulation Process

CFD software uses mathematical tools to solve problems with pre-set equations. CFD is a highly disciplinary subject that indulges into the project area and lies at the interface of physics applied math and computer science[48].

The regular steps for CFD simulation starts with analyzing real problems followed by, numerical modeling, discretization, CFD simulation, post processing and results validation[49]. Each step connects to the next one, the change of any one influences the whole simulation consequently.

The first and the most changeable one is analyzing the real problems. It differs when focusing on different aspects of the same problem. For instance, with different control volumes the analysis would be totally different[50]. What’s more, it’s normally impacted by the change of environment, which leads to another limitation regardless of the properties of fluid or the physical phenomenon.

The second step is the most mathematical one. At this point the analysis of the control volume is done, setting the numerical methods and applying the conservation laws[51]. This step is obviously theoretical and the most important thing is to determine the suitable finite method.

The discretization is about to discretize the continues function applied in CV, which could be N-S equations, to infinite parts with the same intervals ideally[52]. This part is commonly called meshing. With numerical methods, based on these discrete data, one can build a computer program to handle continuous functions. The most important benefit of computer involvement is decreased needed time and increased accuracy.
The next step is to do the simulation according to the meshed model. Adjusting the size and the style of meshing can possibly change the quality of the simulation. It highlights the importance of meshing [53]. After simulation, solutions are shown and post processing is used to visualize parameter such as temperature, speed and pressure. For different parameters, different visualizations are available, which can be contours, vectors and pathlines.

Simulated results mean nothing before they are validated [54]. Validation process is a process in which the simulation is checked with theories or experiments. Since information is lost during discrete process, there always will be some differences between ideal case and real case. This step would be extremely important but may contain a lot of compromises.

In this study, the CFD software used is ANSYS Fluent. It is possible to fulfill the whole process, not only geometry generation, mesh and simulation, but also solver and post processing.
2.2 Finite Methods

Finite methods are used for structure analysis in this project. Finite methods include finite element method, finite volume method, and finite difference method. They are effective in different model structures. In this project, finite volume method is preferred.

2.2.1 Finite Element Method

The finite element method (FEM) is mostly applied to structural analysis of solids [55], but is still suitable for fluid. But one thing needs to be pointed out is that the formulation of FEM requires ensuring a conservative solution[56]. FEM is much more stable than the FVM but it requires more memory and more time to get results.[57]

In the FEM, a weighted residual equation is formed:

\[ R_i = \iiint W_i \ast Q \, dV^e \]

Equation 1. FEM governing equation

Where \( R_i \) is the equation residual at an i-th element, \( Q \) is the conservation equation expressed on an element basis, \( W_i \) is the weight factor, and \( V^e \) is the volume of the element. [58]
2.2.2 Finite Volume Method

The reason why ANSYS Fluent can be so powerful to simulate the models, that it uses finite volume method. The finite volume method (FVM) is one of the most used methods in CFD codes. [59]

Efficient memory usage and high solution speed are the biggest advantages of FVM. FVM is powerful especially for complete problems, high Reynolds number turbulent flows, and source term dominated flows like combustions.

The most important thing in FVM is the partial differential equations to be solved. The N-S equations, combining with mass and energy conservation equations, in one conservative form, are solved over discrete control volumes. While discretization ensures the conservation of fluxes through a particular control volume.[60]

The FVM equation yields governing equations in the form,

\[ \frac{\partial}{\partial t} \iiint Q \, dV + \iint F \, dA = 0 \]

Equation 2. FVM governing equation

Where \( Q \) is the vector of conserved variables, \( F \) is the vector of fluxes , \( V \) is the volume of the certain control volume, and \( A \) is the surface area of the control volume.[61]
2.2.3 Finite Difference Method

The finite difference method (FDM) played an extremely important role in history that is easier to program compared with other methods. But now it is only applied to few specialized codes, which handle complex geometry with high accuracy and efficiency by using embedded boundaries or overlapping grids (with the solution interpolated across each grid). [62]

For FDM method the formula is following:

\[
\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = 0
\]

Equation 3. FDM governing equation

Where Q is the vector of conserved variables, and F, G, and H are the fluxes in the x, y, and z axes respectively.

Finite element method can analysis complex model by mathematic method involved. While finite volume method is aim at physical significance models. Irregular geometries require far more effort by using finite volume method. In this study, finite volume is preferred since physical phrase is very important result.
### TABLE II: COMPARISONS BETWEEN FINITE METHODS

<table>
<thead>
<tr>
<th>Method</th>
<th>Advantages</th>
<th>Disadvantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Finite Element</td>
<td>1. More mathematics involved</td>
<td>1. More mathematics involved - less physical significance</td>
</tr>
<tr>
<td></td>
<td>2. Natural boundary conditions</td>
<td></td>
</tr>
<tr>
<td></td>
<td>(for fluxes)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>3. Master element formulation</td>
<td></td>
</tr>
<tr>
<td></td>
<td>4. Any shaped geometry can be modeled with the same effort</td>
<td></td>
</tr>
<tr>
<td>Finite Volume and Finite Difference</td>
<td>1. Fluxes have more physical significance</td>
<td>1. Irregular geometries require far more effort</td>
</tr>
</tbody>
</table>

#### 2.2.4 FVM vs FEM

It is obvious that FEM and FVM have their own advantages and disadvantages. However, solving the problems of fluid flow along with structural mechanics and heat transfer, it is better to use FEM [63]. While FVM is a specialized method which was designed to overcome the shortcoming of FEM in rigid body problems.
Actually the FVM can be interpreted as a special kind of Galerkin FEM, in which the solution and trial spaces are the same, but different rules of integration approximations are chosen. The most outstanding advantage is that for each volume has two or more nodes, which leads to easily to build continuity or flow variables through the volume under consideration. What's more, since FVM is so robust and powerful that less intensive memory is needed than FEM [64], where each volume is a control volume and it's an individual element.

In this project, physical limits and environment conditions need to be considered, which means FEM will not be suitable. Obviously FVM is used to do the simulation and get the solutions needed to do analysis.

As mentioned, both FVM and FEM are powerful only when they are applied appropriately. We can come to the conclusion that FVM is better suited for this project.

2.3 ANSYS Fluent

2.3.1 Procedure

ANSYS software is one of the world’s most powerful CFD product. This software contains large modelling capabilities and can be used for structural analysis, fluid dynamics, and electronics simulation. In this project, ANSYS Fluent was used to simulate the resin flow in vat. [65]
2.3.2 Geometry

Most businesses consider time and expenses as two of their top critical business issues, especially with prototyping and manufacturing. In 3D printing, speed is critical. Microfluidic grooves are important to lead the fluid especially in CLIP while maintaining the constant elevation speed[68]. Time is saved in this way.

The main problem is the fluid speed and separation force. Designing microgroove patterns on the down window helps to improve the process. Different microgroove designs lead to different fluid speeds. Micro patterns can be fabricated by laser micro-machine, micro milling and lithography.

![Figure 8](image)

*Figure 8. Microgrooves pattern design*

The first model is a radial microgrooves pattern. Liquids flow from surround to the center. When the finished part goes up, the liquid will easily flow to the center. The second model is another radial microgroove pattern with 12 microgrooves instead of 8. By adjusting width and depth, the volume of all the grooves are kept the same as in model 1. By analyzing the results of
model 1 and model 2, relations between radial microgrooves number and flow velocity, and separation force can be done. The cross microgroove is designed differently from radial microgroove. In model 3, the flow will not gather in the center. In radial model, the radius of the center is controlled to avoid resin polymerization.

Three models have different width and have the same volume. With 6 channels, depth is 1cm and width is also 1cm. For 12 channels, depth is 0.5cm and width is also 1cm. With 4 crossing channels, both depth and width are 1cm. The conclusion is obtained by experimental and numerical studies.

Experimental results have shown that resin flow rate and suction pressure varies with different surface texturing. The main aim of different microgroove pattern designs is to alter pressure gradient.

The flow will speed up when the pathway cross section increases. Numerical study can be conducted to analyze the resin flow after computer aided simulation [69]. There is the illustration of window from top view.
2.3.3 Model validation

More and more computer simulations have been utilized to solve fluid dynamic problems[85]. Validations are required to ensure accuracy, which can be done iteratively throughout the model. However, there are two common errors that may cause validation failure. The first is called a model builder risk which means rejecting valid data. The second is called model user’s risk which means accept invalid data.

---

**Figure 9. Illustration of textured window top view**

**Figure 10. Top view and side view of flat pattern**
In this project, validation of solutions is important. To predict the best possible separation force via numerical simulation force and compare it with theoretical results.

Theoretically, when the elevation goes up, the separation force detaches newly cured polymer from the polymerization surface. The separation between resin and the finished part will occur when the platform goes up to a specific height. Separation force can be calculated numerically using three assumptions:

The object part is made of solid cylinder

The bottom window is rigid.

Resin property is constant

Resin flow rate can be derived from N-S equation. The mass conservation equation can give the relation between the resin flow rate and elevation speed.
The pulling up force is

\[ F_z = \frac{3\pi \cdot \mu V}{2 \cdot h^3} \cdot R^4 \]

V is elevation moving speed

R is radius of the cylinder, in our case R is 10mm.

h is the distance from the bottom of the vat and cured layer

As shown in Figure 12, part elevation goes consciously from zero. The acceleration is \(2.96 \times 10^{-5}\) m/s\(^2\).
Figure 13. Part elevation velocity

CLIP method increases 3D printing to 100 times than traditional 3D printing. Velocity keep constant at the speed 0.00015957 m/s from time 0.54 to 1.18. since in this project, separation happens in the early age, so the acceleration keep constant at $2.96\times10^{-5}$ m/s$^2$. 
Simulation has been done from time interval 0.55s-1.2s. Force of flat model and theoretic result is compared.

![Flat pattern simulation compared to theory results](image)

*Figure 14. Flat pattern simulation compared to theory results*

Simulation results can be accepted only when the difference between model simulation result and theoretic within tolerance. Result in Figure 14 shows that the simulation and theoretical calculation are similar to that this model can be used for the other three microgroove designs in simulation.
2.3.3 Mesh

In this project, inflation layer is used. The mesh is inflated along the part geometry boundary. Every model uses nearly 5 million elements.

It is a common choice to create mesh using the Generate Mesh option. The groove is in micro scale and in this project the minimum mesh is 1.06e-5.

Since actual numerical values are non-intuitive and stymie development of domain expertise, therefore interpreting metrics are too difficult. A metric vocabulary should account for desired range of result numerical values. To sum it up, the better the meshing quality the more
accurate the solution. It’s necessary to have a deep understanding of meshing methods and process.[70]

It is possible to use the update option to generate the mesh automatically, to create the relevant mesh files for the projects, and to update the ANSYS Workbench cell that references this mesh.

**Figure 16.** Mesh 8 microgrooves pattern

Since the maximum mesh is 2.1e-3, it is difficult to see the mesh detail. To achieve the accurate result, the mesh of the detect area should be very fine.
Figure 17. Mesh with cross microgrooves pattern

Figure 17 shows the mesh in cross microgroove patterns. The minimul mesh size is similar with radial microgroove patterns.

Figure 18. Inflation outside boundary
The cut cell meshing was used in the project, and it is convenient because it is easy to get a predominantly hexa mesh with inflation layers to capture the viscous boundary effects.

The model is divided into two assembly parts. Figure 12 shows the inflation outside the boundary. The mesh size is relatively big since this is not directly detected.

![Figure 19. Dead zone](image)

The first part inside boundary is the dead zone. Like shown in the figure 13. It is the crucial area for separation force and flow rate. Resin inside the dead zone should be carefully monitored.
Cut cell mesh distinguishes the crucial part from other assembly parts. What’s more the solver dealing with this kind of mesh converges more quickly and accurately than with tetra/prism meshes.

### TABLE III: SUMMARIZES THE ADVANTAGES AND DISADVANTAGES OF CUT CELL MESH

<table>
<thead>
<tr>
<th>Advantages</th>
<th>Disadvantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. They can be generated quickly</td>
<td>Only certain codes such as cell-centered codes can handle them (e.g. this means that Fluent can use them but not CFX)</td>
</tr>
<tr>
<td>2. They result in predominantly high quality hexahedral elements</td>
<td></td>
</tr>
<tr>
<td>3. Usually be solver friendly, meaning that solution convergence is generally faster.</td>
<td></td>
</tr>
<tr>
<td>4. They can be adapted (refined/coarsened) very quickly based on a predefined criterion (Fluent can do this during the solution procedure)</td>
<td></td>
</tr>
</tbody>
</table>

In order to generate them in ANSYS, either ANSYS Meshing or Fluent (with and without T grid meshing model) is easy to operate. However the meshing quality highly depends on the size of cut cell [71]. McDaniel showed a correlation of mesh quality with solution accuracy with the caveat that a well resolved mesh can have poor quality and still produce a good answer.

The result is much more accurate with finer mesh. Using as many grid points as possible, in many cases, resolution trumps quality. However, the practical matter of minimizing compute time by using the minimum number of points, means that quality still will be important, and that’s what Thornburg called an optimum mesh. With all the above procedures completed, the next step is to set up the physical part of the model.

### 2.3.4 Mesh independent study

In order to have more confidence in our results, we should make sure that the result from CFD is independent of the mesh resolution.
The separation force derived from the model method and simulation is compared. The error should not exceed 0.01%.

The error of mesh number 2 million is 2% percent which is unacceptable. In Figure 20, simulation error can be reduced by increasing mesh number.

Refine the mesh globally so that the geometry has finer meshes. 3 million cells were conducted, and the error reduced to 0.2%. Solution is changing because of the mesh resolution is not yet independent of mesh number.

The mesh independent solution is achieved until the solution is stable with the refinement mesh. The mesh size reach to 4 million the error is in tolerance (0.1%) by increasing the mesh size further, error goes to 0%. This indicates that we have reached a solution value in dependent with mesh resolution.
To ensure the result accuracy and saving calculate time 5 million mesh size number was chosen as an optimal value.

2.3.4 Set Up of the Simulation in ANSYS Fluent

After creating a geometrical computational mesh model, the next step is to set up a CFD analysis using ANSYS Fluent, then review the list of files generated by ANSYS Workbench [72]. The geometry and mesh are defined in a static way without any motion. In real simulation, all the factors vary with time.

The specific working conditions should be defined before starting calculation. Examples of conditions are inlet fluid velocity, pressure, and capacity. The model choice is defined in the specific way to fit the specific problem.

2.3.5 Check Mesh

The main reason for checking the mesh is to ensure that the minimum volume is positive. ANSYS cannot start working if the minimum volume is negative, and mesh quality can also be analyzed during this procedure.

Mesh is used to analyze objects and the higher mesh quality, the more accurate the solutions are. It is interesting to notice that meshing helps to simulate but during the mesh, for each CV information lost during discrete.
Mesh size is the number of meshes in the specific area. Commonly the geometry is meshed into pieces. In some conditions, especially when the geometry is quite small, the result will be negative. A negative result means the particle is smaller than the mesh size [73]. In this project the original model size has been done within 1µm meshes, which leads to negative results, and that is why mesh size has been enlarged.

Mesh cell can be classified in two ways, two-dimensional and three dimensional. The main problem and solver capabilities are the main factors on which choices depend.[74]

![Basic two-dimensional Cell shapes](image)

*Figure 21. Type of mesh cells*

Triangle cell shape is the most easy and simple mesh shape. It is easy and quick to create triangle cell based mesh surface. Quadrilateral is most used in structure grids. It requires smooth surface not concave or convex.
Three dimensional mesh cell models are the combination of triangle and quadrilateral. The most used case is tetrahedron. Pyramid is effective in triangle, square and especially in hybrid meshes. Triangular prism has complex surface and shape for resolving boundary layer. Hexahedron is also known as hex or brick. Compared to other cell mesh shapes, hexahedron has the highest accuracy.

Classification of grids: Unstructured grid has cells arranged randomly, without index and constraints. It can be used in all the geometries.
Structured grids: The structure is defined in 3 indexes, ijk. The mesh can be represented in a single block. In Figure 24 is the logical representation, all kinds of shape can be described in this logical representation. Computer will demonstrate how much force applied on each single box.

![Figure 24. Logical representation](image)

It is easy to gain the information because single block meshes can be cornered into 180 degrees. Another kind of grid is hybrid grid which contains grids mix [53]. Since real models are complex and uniform, hybrid grid can always have the best accuracy than any other combination.

In this case, pressure near the hole and speed in grooves changes much faster than surrounding area. Therefore, it’s necessary to modify the mesh cell around the hole otherwise big error occur in the solutions.
For example, circular hole in the center with hybrid shown above the case mentioned before. The mesh example in our model is as follow.

![Mesh with 8 microgrooves pattern](image)

*Figure 25. Mesh with 8 microgrooves pattern*

The area under the part elevation is much finer compare with the surround part where is indicated by red color. The following volume statics give more details about the volume. Here it’s clear that minimum and maximum volume and total volume and then it will check area static.

What the check mesh does first is to check the face and node matches. For example, a quadrilateral mesh cell in two-dimension should have four faces and four nodes. In three dimension a hexahedral cell must have six faces and eight nodes. If in ANSYS Fluent detects it is not the case, it will generate an automatic warning.
Next problem will be face handedness and face node order. Since quiet small mesh is required for geometry like this, the order in the mesh geometry is important. Most of time the right handed faces are chosen, with right and clear node order. If this rule breaks in mesh generation, mesh will generate interface. When force applied on geometry, especially deformation happens, ANSYS Fluent is impossible to move and calculate the characters [75].

The last part is check according to axis, the node and face should be strictly upper X axis. Since our model is symmetric, periodic boundaries are implied. The main track here is the periodic angles. The common mistake is define the periodic angle with wrong range.

Finally, with all the correct specify, ANSYS Fluent can go on and continue to setup. Generally, the regular mesh is more acceptable after mesh generation. Node is regular on every part of the geometry. In the element volume, too large gradient of variable can also cause large error which should be avoided in this research.
2.3.6 Solver

2.3.6.1 Materials for the CFD Simulation

PDMS is short for polydimethylsiloxane that belongs to a group of polymeric organic silicon compounds that are commonly referred to as silicones. The down window is made of PDMS. The curves are made on PDMS membrane.

PDMS is commonly used in microfluidics chips for flow delivery[76]. One of the advantage is that the milling microgroove on PDMS material can be nanometer diameter channel. It is basically used on glass, silicon or polymer. It is mostly used in normal technique of photolithography or lithography. It is also widely used in biomedical system.

Another very important character is that, PDMS has an anti-foaming function. Due to the stable chemical and mechanical structure, PDMS will not have reaction with the resin in curve.

2.3.6.2 Liquid Property

The original resin is photosensitive polymer liquid. It absorbs UV light energy and became solidified. Chemical reactions happen during the solidification. Resin cell structure and chemical and mechanical properties changes substance. These materials are eco-friendly and therefore popular in recent history since they can work without any volatile solvent in most of the industrial processes [77]. Type of resin are chartered by the property, such as strength and durability.
Viscosity is another important characteristic. High viscosity means strong shear stress and tensile stress. This need higher requirement of support structure and pulling force [78]. Building speed will be very slow. The building time only refer to the part build time, all the post process are not included in build time. The post process such like removing the structure also speed a lot of time.

So, with extremely high viscosity liquid should not be used at this point of view.

Most condition, high viscosity resin has faster ability to solidify. There should be a balance between polymerization speed and viscosity.

Figure 27. Changes in chemical properties Photopolymer at Wikipedia
Acrylic resin is one kind of thermoplastic substances. It is most often used in automotive parts, medical device and construction. Acrylic resin has high flow speed, high mechanical property and short UV post photopolymer[79].

### 2.3.6.3 Boundary Conditions for the CFD Analysis

Boundary conditions are important in this project. It is used to define the liquid flow state. It is also important for Naiver-Stokes equation and continuity equation.

Two main works in define boundary condition is define the boundary locations and specify boundary property. If possible, select boundary location such that the flow either goes in or out. Minimize grid skew near the boundary [80], otherwise error can be found in the very early. What need to do is identifying the property in the boundaries, include inlets location, inlets velocity, and symmetry. The result will be significantly influenced by poor defined boundary condition.

In this project, velocity inlet is zero. Since multiple inlets are presented in this case, therefore mass conversation should be satisfied. Pressure inlet: Because of incompressible flows, so total pressure is the sum of static pressure and kinetic pressure.
2.3.6.4 Moving Boundary

The standard window cannot meet with all user’s need, thus user can write their own function by C language. ANSYS Fluent can link C language dynamically language can define flexible boundary conditions, source terms, material property, etc. In this project, the define of the elevation’s motion is needed. In ANSYS there’s user defined functions (UDF) help users to apply functions they need. Since the motion is simple fixing 4 points, which is doable by UDF easily.

Starting point at time zero. The elevation start moving up with constant acceleration. When the speed reach to the $1.6e-5$ m/s, keep in high speed for 0.64 seconds, the velocity decreasing at a constant deceleration. Until the speed reach to the zero point. The following codes is what used in this project. Other characters can be calculated after speed of continuous elevation is well done.

2.3.6.5 Run Calculation

This is the last step for calculating. In CFD the solution is not calculated all at one time[81], the result will be done iteratively. Convergence is the sate computer reached a stable point. From liquid point of view, the flow has reached a state of equilibrium.

As the solver run through many iterations of solution, the convergence plot show that the fluid has reached the steady state. In our project, the resin flow is transitioned [82].Obviously, it
cannot reach steady state condition. The way to find stop the iteration and get the result is to define the iteration times.

A best result achieved when it is necessary that the result will be convergent after calculation. Within 50-100 iterations if the result cannot be convergent, what need to do is to decrease time step.

In this project, iteration were defined to be 2000. The plot show it is perfectly convergence. It take 2 minutes to get the result. Define the number of iterations and report interval, and profile update intervals, then start calculation. With a starting number, the convergent speed and related time are presented[83] It is necessary that the result will be convergent after calculation. Within 5-10 iterations if the result cannot be convergent, one needs to decrease time step. In the same case, the more iterations the more accurate.

### 2.3.7 Post-Processing

After Fluent finished the iteration, results can be displayed in different ways. Depend on the fluid character and analyze methods, different display methods can give more intuitionistic results according to the specific case.
2.3.7.1 Contours

The most common post-process display is contour. It is also very useful for this project. It is clear to see the effects of meshing and numerically on a simulation solution. For instance, one can observe some asymmetry in the results by similarly plotting Axial Velocity.

The color in contour can help to distinguish the values. The most and least pressure distributed is clear. Pressure distribution is obvious. The red part has the highest pressure as indicated in the graph. With surface spread out, the pressure is also decreased. In this project results of pressure and velocity will be shown in contour display in chapter 3.

In this project results of pressure and velocity will be shown in contour display in chapter 3. The pressure and velocity difference can be seen from the darkness of contour color

2.3.7.2 Display the Vectors

Vector display is useful in showing velocity and relative velocity. In surface window, it’s possible to either choose a single surface or the entire surface.

The result can be analyzed in two ways. Firstly is the length of the vector, the area with longer vector indicates higher velocity. Secondly, similarly the result in the contour displays, the red color indicates highest value. Usually, vector is too small to be shown in large figures.
2.3.7.3 Generating path lines and Particle tracking Display

Path line and particle tracking are two graphic displays of mixed phases. They give continuous representation of the second phase velocity varying with time. It will not be used in this project since the resin is considered homogenous. The liquid is single phase.
CHAPTER 3

RESULTS

Three models compared with flat pattern have been designed to study the relationships of microgroove pattern design with resin flow rate and separation force.

3.1 Simulation Results

Separation force is a critical factor to enlarge the capability of 3D printing. Large cross section object needs high separation force. Low pressure occurs in liquid that can cause cavitation. Cavitation usually occurs when a liquid is subjected to rapid changes of pressure that cause the formation of cavities where the pressure is relatively low.

![Diagram of phase diagram with pressure and temperature showing the formation of cavities](image)

*Figure 28. Formation of cavities*

In order to fabricate objects with large cross section, separation force should be reduced.
3 textured window compared with the flat pattern is analyzed.

Figure 29. Force of 4 different textured surface

3 models have been simulated with the same setup, moving up with the same velocity and the separation forces are shown in the figure above. As demonstrated in model validation,

\[ F_z = \frac{3\pi \cdot \mu V}{2 \cdot h^3} \cdot R^4 \]

The separation force increases with the velocity accelerations. Even when velocity is constant, the changes of separation force are different. Obviously flat pattern as showed in graph with red
color has maximum force and keep still until velocity decreased. However 3 models perform great as decreasing separation force because of textured windows increase the velocity of flow refilling the rap and make it easier to move up.

8 radial microgrooves pattern helps the resin speed up. With curves designed on the window, the fluid flow in to the center easily. From theoretical point of view, force is a function of velocity, the higher the speed, the smaller the force.

12 radial microgroove pattern helps to increases the maximum force to 23.58N. Compared with 8 radial microgroove patterns, this one is more efficient. It is more suitable for 3D manufacturing production with large cross section.

Cross microgroove pattern differs from radial microgroove patterns. In this case, liquid will not go to the center of the window. The maximum force of 26.08N, it is higher than the numerical one, but it is not the first choice since radial microgroove pattern has the best performance.

In sum, there is an obvious negative correlation between separation force and curve design. A better curve design can be found in the future to increase the efficiency of 3D printers.

3.2 Pressure Analysis

Surface force is includes pressure force and stress force. Pressure force is normal to the cross sectional area and stress force is tangential to the area. Pressure analysis is important for calculating the relative force inside vat. Integrating pressure over the area gets the normal separation force.
The pressure difference between the evacuated volume and the surrounding atmosphere drives fluid upward. To keep the printing process going continuously, the liquid should never be separated with the finished part. So the maximum pressure is a critical problem for printing process.

Pressure differs from center to the edges. The dark red indicates the maximum pressure is 6.09e-01 Pa. The blue color indicates the minimum pressure is -1.09e+01 Pa. This criterion is fixed for all the simulations.
Figure 32. Flat surface pressure distribution at time 0.6s (left) and 1.0s (right)

Figure 32 shows the pressure distribution of flat pattern surface. Maximum pressure happens around the finished part. Pressure decreases from the outside to the center. The minimum pressure happens in the center of the cylinder. Flat pattern have high vacuum pressure indicated large blue area. Pressure increased after 0.4s since velocity keep constant during this time, vacuum is highly filled by liquid.

Figure 33. Cross pattern pressure distribution at time 0.6s (Left) and 1.0s (Right)
In cross pattern, the blue area reduced. The vacuum pressure increased.

Figure 34.12 Radial pattern pressure distribution at time 0.6s (Left) and 1.0s (Right)

The distribution of pressure in 12 radial pattern is similar with 8 radial pattern. 12 microgrooves radial pattern can better alter the pressure gradient since the value is reduced largely. According to the figure, pressure increases from center to the edges. Maximum pressure at the edge. Pressure in the center is reduced to 12 radial pattern has minimum pressure at time 1.0 second we can hardly see the blue area.
After finishing one layer, resin needs time to return to static condition. The pressure distribution can be seen in the graphic above, under the elevator pressure is relatively big especially insider the microgrooves, where the maximum pressure occurs. Similar to the last one, high pressure is around finished part.

For cross microgroove pattern, pressure in the center area is up to -4.41 Pa. It’s much higher than the pressure in radial microgroove pattern.

At time 0.5s, the maximum pressure reaches 33.3Pa. The pressure in the center reaches -18.7Pa which hugely increased the separation force.

To sum it up, the 12 microgroove radial pattern has better performance compare to 8 grooves radial pattern and cross microgroove pattern, pressure is relatively small which leads to a relatively small force. All of these will improve 3D printers’ performance.
3.3 Velocity character

Velocity of finished part increase at constant acceleration, keep constant at maximum speed for few seconds then velocity decrease to zero. The fluid of resin in vat is analyzed.

The velocity frame of reference is made for 4 patterns.

Maximum velocity is 2.74e-03 m/s.

Minimum velocity is 1.61e-18 m/s.

Velocity is the key factor for 3D printer’s efficiency. There are many factors that influences the speed of the printing process, for example slicing velocity and sintering speed. Currently there are three speed ranges. First range up to 50mm/s, second range can be nearly 100mm/s, the fastest printer work at 250mm/s. A faster speed 3D printer is desirable.

High resin velocity is desired. The liquid under finished part is no slip condition.

Velocity taken from the face 0.05m from the upper surface.
The fluid around the finished part is the most stressed and it has the highest speed as shown in red color. With 8 microgrooves, pressure is relatively low in the center. The maximum velocity is $1.63 \times 10^{-6} \text{m/s}$ which is quite high.
With increased groove number, the velocity increases as well. Comparing with the first model, this model has more significant efficiency.

Contrasting to the radial microgroove patterns, biggest velocity concentrates in the central area.

It is important to notice that after one layer finished, a short time is needed for resin to return to static condition. The higher speed, the more time needed to return to static condition which will highly decreases efficiency.
Chapter 4

CONCLUSION

3D printing can do what traditional manufacturing cannot do. This technique bridges commercialization where robust devices can be more easily shared and disseminated [86].

3D printing will be the future with endless opportunities, it has changed our manufacturing method and it will continually impact the way people live and work. However, commercial productions made by 3D printers, for home or personal use, are not so widely used so far. A low cost and high speed, high resolution 3D printer is the aim.

CLIP is one of the latest technologies of 3D printing. In CLIP both light and oxygen are used as tools to build objects from a vat of resin[87]. The way it builds objects is still layer by layer, similar to the traditional one, but the main difference is that in this new technology the mechanical steps that were used to build the model layers are eliminated, which will save a lot of time.

Although CLIP is a technology producing objects with high speed and high quality compared with traditional methods [88], there are still improvements needed, such as increasing speed. This project offers a new method to speed up the flow rate. It will break the limit of 3D printing in fabricating large cross section objects and shorten building time significantly. The brief idea is to find the fastest available speed through microgrooves.
With different window designs, resin flow rate changes significantly. Among three models illustrated, the 12 microgrooves radial pattern has the best performance. However better designs can be made to help speed up the resin flow rate, but in this case radial shape is the best.

An interesting development occurred recently. As 3D printing is actually based on 2D printing, 4D printer technology is developed based on 3D printing which adds the time component [89]. This will be extremely useful, which is the main reason that 4D printing is becoming more attractive. Because the production shape can be changed from one dimension to two dimensions or three dimensions, there are infinite possibilities. The project demonstrated, the new technologies make for better and easier living.
CITED LITERATURE


CITED LITERATURE (continued)


52. Moon, W.: The efficacy of maxillary protraction protocols with the micro-implant-assisted rapid palatal expander (MARPE) and the novel N2 mini-implant-a finite element study. Prog Orthod, 2015. 16: p. 16.


70. Free tool for MeSH grids. CILIP Update, 2016: p. 16-16.


VITA

NAME Tiantian Ma

EDUCATION

Master of Science in Mechanical Engineering, University of Illinois at Chicago, August 2016, USA

Master of Science in Mechanical Engineering, Dec 2016, Polytechnic of Turin, Italy

Bachelor’s Degree in Mechanical Engineering Jul 2014, Polytechnic of Turin, Italy

LANGUAGE SKILLS

Chinese Native speaker

English Full working proficiency

Italian Communication skills

SCHOLARSHIPS

Fall 2011-2014 Italian national education scholarship for bachelor student

Fall 2014-2016 Italian national education scholarship for master student

Fall 2014 Italian scholarship for final project (thesis) at UIC
Sep 2015-Present  Surface modification for faster 3D printers

   Built 3D model in Solid works; analyzed different surface modification
   with ANSYS FLUENT Fabricated the best resin flow surface pattern.
   Improved the 3D printing efficiency

Feb –July 2015  Hydraulic pneumatic control system design

   As a hydraulic lab assistant, had a real touch understanding and operations
   of pumps, valves.

Feb-July 2015  Factory based thermal and hydraulic design

   Invented a district heating network with radiative heat transfer and HAVC
   layout Designed heat recovery steam generator, heat exchangers and pump
   system

Sep 2014-Jan 2015 Designed best efficiency of gearbox

   Worm & worm gears and Rack & pinion gears design and analysis,
   combining bearings, belts and bolts, had an all-around understanding of
   industrial machines.