CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Nowadays injection molding is probably the most important method of processing of consumer and industrial goods, and is performed everywhere in the world. The developing of injection molding becomes a competition from day to day. This process now integrated with computer control make the production better in quality and better quantity.

In designing the mold for injection molding, the accuracy in making mold very important in order to reduce and also to make sure that the mold broke easily. Before this, the mold designer used manual analysis to the mould. But now, there is software that can simulate the analysis of the mold that wants to develop.

Clearly, more manufactures are using computational and analytical techniques to reduced design time and cost while significantly improving yield and quality. By using plastics flow simulation products, the determination of manufacturability of part in the early design stages and avoids potential downstream problems which can lead production delays and cost overruns. Some of the materials that have been used are very expensive. Therefore, less time on the production floor working through a problem saves labor and material costs. These days, simulation software can accurately predict the fill patterns of any part. This allows for quick simulations of gate placements and helps finding the optimal location. Problem that can be avoided by performing flow analysis early in the design stages are, sink mark, air traps, shrinkages, and blush and flow marks.
1.2 OBJECTIVES

For this project, there are 3 main objectives to achieve the target. The objectives are:

(i) Investigate the gate mechanism effect on injection molding parameters and defects of book tray.
(ii) Design and proposed gate mechanism according to the results analysis.

1.3 PROJECT SCOPES

One of the most important parts in a project is the project scope. In order to get the best result, the scopes are:

(i) Analyze gate mechanism consists of number of gates, location and size of the gates.
(ii) Using the Moldflow Plastic Insight (MPI 5.0) as the main software to analyze.
(iii) Comparison selected parameters and defects which are volumetric shrinkages, air traps, and sink index on each gate mechanism.

1.4 PROBLEM STATEMENT

The trends of producing a plastics product in injection molding industries are recently changing from traditional method to using the FEA analysis. For injection molding industries, time and cost is very important aspects to consider because these two aspects will directly related to the profits at a company. The next issue to consider is the number and location of the gate. In some cases, the product designers will indicate how much and where they believe the gate should be. Number and location of gates must be selected because the function and strength of the product depend on that factor.
The filling of cavity slow or impossible to fill the cavity full before it freezes. This is because of the gate that have been is too small. However, too large gate can make the gate and the product that been joining hard to break and will make mark in the product. In order to get the best parameter for the injection molding process, plastics have been waste. Through the experiment, operator will use large amount of plastics material to get the possibly parameters to setup the machine.