DESIGN AND ANALYSIS OF TWO PLATE MOLD INSERT FOR DOG BONE SPECIMEN

MOHD ALIFF BIN MOHD RADZI

BACHELOR OF MECHANICAL ENGINEERING
UNIVERSITY MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS♦

JUDUL: **DESIGN AND ANALYSIS OF TWO PLATE MOLD INSERT FOR DOG BONE SPECIMEN**

Saya **MOHD ALIFF BIN MOHD RADZI (860809-11-5081)**
(HURUF BESAR)

mengaku membenarkan tesis (Sarjana Muda/~~Sarjana~~ /~~Doktor Falsafah~~)* ini disimpan di Perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Malaysia Pahang (UMP).
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

☐	**SULIT**	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
☐	**TERHAD**	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
√	**TIDAK TERHAD**	

Disahkan oleh:

(TANDATANGAN PENULIS) (TANDATANGAN PENYELIA)

Alamat Tetap:

C202,TAMAN GEMILANG
BANDAR CHENEH BAHARU
24000 KEMAMAN
TERENGGANU

EN. ASNUL HADI BIN AHMAD
(Nama Penyelia)

Tarikh: **23 NOVEMBER 2009** Tarikh: : **23 NOVEMBER 2009**

CATATAN:
* Potong yang tidak berkenaan.
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu dikelaskan sebagai atau TERHAD.
♦ Tesis dimaksudkan sebagai tesis bagi Ijazah doktor Falsafah dan Sarjana secara Penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled "Design and Analysis of Two Plate Mold Insert for Dog Bone Specimen "is written by Mohd Aliff Bin Mohd Radzi. I have examined the final copy of this project and in my opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in partial fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering with Manufacturing Engineering.

Examiner	Signature
Ms. Nurul Shahida Mohd Shahlahim	....

DESIGN AND ANALYSIS OF TWO PLATE MOLD INSERT FOR DOG BONE SPECIMEN

MOHD ALIFF BIN MOHD RADZI

Thesis submitted in fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Manufacturing Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Manufacturing Engineering.

Signature
Name of Supervisor: EN. ASNUL HADI BIN AHMAD
Date:

STUDENT'S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature
Name of Student: MOHD ALIFF BIN MOHD RADZI

ID Number: ME 07006

Date:

Special to my beloved Ma & Abah,
Mohd Radzi Bin Mohd Datar and
Tuan Fatimah Binti Tuan Hamzah

ACKNOWLEDGEMENTS

The Author grateful and would like to express sincere gratitude to supervisor who is En. Asnul Hadi Bin Ahmad for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He has always impressed me with his outstanding professional conduct, his strong conviction for science, and his belief that a Degree of Bachelor program is only a start of a life-long learning experience. The Author appreciates his consistent support from the first day The Author applied to graduate program to these concluding moments. The Author is truly grateful for his progressive vision about my training in science, his tolerance of my naïve mistakes, and his commitment to my future career. The Author also sincerely thanks for the time spent proofreading and correcting all the mistakes.

Sincerely thanks go to all colleges and members of the staff of the Faculty Mechanical Engineering, UMP, who helped in many ways and made The Author stay at UMP pleasant and unforgettable. Many special thanks go to member molding research group for their excellent co-operation, inspirations and supports during this study.

The Author acknowledges sincere indebtedness and gratitude to The Author parents for their love, dream and sacrifice throughout their life. The Author also feel grateful to Pok Tar where helps and allows The Author done this study property with help and prays to make this work possible with blessing of Allah S.W.T. The Author cannot find the appropriate words that could properly describe the appreciation for their devotion, support and faith in ability to attain the goals. Special thanks should be given to final year students members. The Author would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

ABSTRACT

This thesis deals with durability assessment for design and analysis of two plate mold insert for dog bone specimen using selected sample. The objective of this thesis is to design two plate mold insert and analyze the material flow of the selected product. The thesis describes the material flow analysis to analyzed the material flow in the part and identify the best gate locations of the components. Acrylonitrile butadiene styrene (ABS) materials were studied in this thesis as sample of material and it is commonly used in industry and lab. The structural three-dimensional solid modelling of two plate mold was developed using the computer-aided drawing software. The strategy of analysis model was developed. The material flow and gate location analysis was then performed using Moldflow Plastic Insight software. The analysis model of the components was analyzed using the two cavities with cold runner design. Finally, the orientation skin for 21 number of different gate location is analyzed. From the results, it is observed that the analysis using Moldflow material flow analysis show the different orientation skin for different gate locations. The gate locations is constrain at the middle of the part where the area that would be test for the specimen. The orientation skin at the both sides of the part is same but only change the length of runner. The increase of length of runner will increase the projected area, the clamping force, filling time and the material waste. So that, the best locations of gate is at the nearest or as short as it can for runner length to be design. But, the first priority is the orientation skin for the material flow. The orientation skin should be same at the middle of part which is parallel to the strength that would be applied on testing. The final design was done in solid modelling and details drawing using SolidWorks software. The results can also significantly reduce the cost and time to fabricate the insert. That is also can be a reference for the future works of fabrications.

ABSTRAK

Tesis ini membincangkan dengan taksiran ketahanan untuk rekaan dan analisis dua plat acuan dengan bentuk mengikut specimen tulang anjing menggunakan saiz yang terpilih. Objektif tesis ini ialah untuk mereka bentuk dua plat acuan dan analisa aliran bahan bagi produk yang ditetapkan. Tesis ini menggambarkan analisis aliran bahan yang dianalisis aliran masuk ke bahagian dan mengenal pasti pintu terbaik dimana lokasi pada bahagian itu. Acrylonitryl butadiene styrene (ABS) bahan yang telah dipilih untuk dianalisa dalam tesis ini sebagai sampel bahan dan ia biasanya digunakan dalam industri dan makmal. Pemodelan dalam bentuk tiga dimensi struktur dua plat acuan dilakukan menggunakan lukisan dan gambaran menggunakan komputer perisian. Strategi model analisis ini telah dibangunkan. Aliran bahan dan analisis lokasi pintu adalah dilakukan menggunakan perisian Moldflow Plastic Insight. Model analisis bahagian-bahagian itu dianalisis menggunakan dua buah lubang berbentuk specimen tlang anjing yang menggunakan reka bentuk pelari sejuk. Akhirnya, kulit orientasi untuk 21 lokasi pintu yang berbeza dianalisa. Daripada keputusan-keputusan itu, ia diperhatikan yang analisis yang menggunakan aliran bahan Moldflow analisa menunjukkan orientasi berbeza kulit untuk lokasi-lokasi pintu yang berbeza. Lokasi-lokasi pintu dielakkan di bahagian tengah di mana kawasan yang akan digunakan untuk meguji specimen tersebut. Kulit orientasi pada kedua-dua bahagian adalah sama tetapi hanya perbezaan dan penambahan panjang pelari yang digunakan. Peningkatan panjang pelari akan meningkatkan luas terunjur, daya mencengkam, masa memenuh dan sisa bahan. Untuk lokasi terbaik pintu itu adalah pada jarak terdekat atau sependek ia boleh untuk panjang pelari direka bentuk. Tetapi, keutamaan pertama adalah pada bentuk kulit orientasi untuk aliran bahan didalam produk. Kulit orientasi sepatutnya sama di pertengahan bahagian yang mana selari dengan arah untuk daya kekuatan yang akan dikenakan di dalam ujian spesimen. Tujuan terakhir adalah dengan pemodelan pejal dan lukisan lengkap menggunakan perisian SolidWorks. Keputusan-keputusan itu boleh juga nyata sekali mengurangkan kos dan masa untuk mereka sisipan. Iaitu juga boleh jadi suatu rujukan untuk kerja-kerja masa depan pemesinan sebenar.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

ABS	Acrylonitrile-butadience-stryene
ASTM	American Society for Testing and Materials
C	Celcius
CAD	Computer-aided drafting
CAE	Computer-aided engineering
D	Diameter
F	Fahrenheit
FKM	Fakulti Kejuruteraan Mekanikal/ Faculty of Mechanical Engineering
HDPE	High Density Polyethylene
L	Length
mm	Millimeters
MPA	Moldflow Plastic Adviser
MPa	Mega Pascal
MPI	Moldflow Plastic Insight
PC	Polycarbonate
PET	Polyethylene terephthalate
PL	Parting Line
POM	Acetal
PP	Polypropylene
PS	Polystyrene
PSM	Projek Sarjana Muda
SAE	Society of Automotive Engineers

SAN	Styrene-Acrylonitrile
t	Thickness
W	Width
3D	Three Dimensions

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

Plastic injection molding is one of the most important polymer processing operations in the plastic industry today. The plastic industry that is injection molding process and involving manufacturing has high growth potential. Most of the products are made from plastic material. The aim of this project is to design and analysis of two plate mold insert for Dog Bone tensile test specimen. This work is concerned with the design, analysis, and simulation two plate molds insert for plastic injection molding.

In designing mold insert for injection molding, the accuracy in making mold is very important in order to reduce the cost also to prevent mold easily broken. Previously, the engineer or designer of mold used manual analysis to the mold. But, nowadays the technology becomes more advances and the analysis replaced by software that can simulate the mold that want to fabricate.

The insert for two plate mold will design with two cavities of *Dog Bone* tensile test specimen. The designs are completely investigate on number of cavity, number of gate, runner system, and gate mechanism for dog bone tensile test specimen insert. The design is focus on the application of product that has some limitation when propose the design.

This project concern with two software which is Computer Aided Design (CAD) and Computer Aided Engineering (CAE) Moldflow Plastic Insight (MPI) . The design two plate mold insert start with draw the dog bone specimen which is follow the

American Society Test Method (ASTM) specification and size. Then one sample size of dog bone specimen is choosing for the sample of study. This drawing stage is by using the CAD software which is SolidWorks. The mold based drawing also require with suitable size for twin cavities of dog bone specimen. This mold based is follow as the original dimension in the lab.

Then, the simulation and analysis of the design will investigate by using CAE software which is Moldflow. This software is to analyze the material flow and to determine the best gate location for desire mold insert. The analysis is run to the drawing of dog bone specimen where the design as their shape and dimensions. The analysis is limitation on one sample of selected material which is *Acrylonitrile-butadience-stryene* (ABS) and the gate could not be place at the test area on the specimen. This software will show the effect of gate location with desire runner size to the orientation skin on product (dog bone specimen) and the other parameter such as the fill time, and the defect if appear. This software is as a guide to change and choose the best and correct parameter to the mold insert design. This step is the normally way to eliminate the rework cost and time as all the possible errors are being eliminated before it actually occurs in actual production or machining process.

The final result of analysis will conclude as final design with details dimensions and size of mold insert. All the finding will draw in the SolidWorks as a completed design includes 3D modeling, details dimensions and suitable and recommended parameter for dog bone tensile test specimen.

1.2 PROBLEMS STATEMENTS

Mold is an expensive device that used for plastics process. The mold design should be precise, accurate and effective in dimensions to reduce the cost and possible defect that can occur on product. This project decided based on the problems occur which are;

- Parts has a defect caused by the Injection Mold Design

- More than three time mold testing require to balancing material flow into the part especially in multi-cavities and family mold.
- High costs on mold insert fabrication.
- Precise and accurate dimensions of mold insert design require as desire product shape.
- The gate location is effected to reduce the defect on product or parts.

All the problems that were occurring during designing process should be solving properly. Previously, the engineer and designer solve all the problem and difficulties by get the reference or guidance from the past experience of earlier mold design and by experienced engineers. The technology today has helps the engineer to solve all the problems. This is come by using computer software where help the engineer easily change or determine the optimum gate size, gate location and so on. This is directly will reduce the time and cost of production.

1.3 PROJECT OBJECTIVE

The objectives of this project are:

1. To design Two Plate Molds insert for injection molding process.
2. To determine the best material flow for twin cavities to reduce the friction flow during injections.
3. To determine the best gate location for less defect to test specimen.

1.4 PROJECT SCOPES

The Scopes for these projects generally focus on designing using CAD software (SolidWorks) and analysis using CAE software (Moldflow) which is to find the best design of mold insert as steps before fabrication. Firstly, the focus of analysis is the dog bone specimen. This specimen used as a tensile test specimen where follows the standard of American Standard Test Methods (ASTM). The sample size and type was choosing based on that standard of ASTM.

Next, this study used the specify CAE software for analysis the material flow which is Moldflow Plastic Insight (MPI). MPI with 5.0 versions used to determine the best gate location for the dog bone specimen. This software will show how the material flows into the parts in order to determine the best location for the design.

This study was focus on design the best gate location for twin cavities of dog bone specimen insert. This design is to reduce the possibility defects that can occur and to have the effective design with less of product defect after mold insert fabrication. All the design was focused for two plate mold as the title of this study.

Lastly, the analysis was performed by using selected engineering material which is Acrylonitrile Butadiene Styrene (ABS). This material selected for one sample of material for analysis and that is suitable for many applications for engineering works.

1.5 PROJECT PLAN

Gant chart for the project plan can be referring to appendix A1 for Gant Chart PSM I and appendix A2 for Gant Chart PSM II.

This “Projek Sarjana Muda” (PSM) consists of two parts which are PSM 1 and PSM 2. That is separate for first semester with PSM 1 and continues with PSM 2 on second semester. PSM1: At beginning with receives the title of project from supervisor and supervisor gives briefly explanations about that PSM title given. Supervisor give briefly guide to make the schedule or task planning for the whole works for PSM. Schedule management is needed for this project to make sure the project running in progress. Then, discussion with supervisor start focused on the project title about the objective, problem statement, and scope of project. Next, the project progress continue by start finding the information data and all related information to the project title by reverse engineering study and literature review from web sites, journal, reference books, supervisor and other relevant academic material that related to this project. To get the clear view and very understand about the project, it is require studying more on material that related to the project topic and spend more than two week to make a literature review. The process of literature review in continuously move from one thing to another things. This is continuously until the project arrived to chapter 4 which is result of the project. Every week and continuously, improvement of knowledge is needed to make sure this project will be performing very well.

Continuing the progress, all the literature review related need to collect and get focused to reduce the scopes of works. The literature review requires the information and material about injection mold, mold components, injection mold design, mold based, gating system, runner system, and the software that is required for this project. The progresses continue with product or parts selection as a sample of product study for this project. The sample of product which is Dog Bone tensile test specimen was choosing. This product, scopes to standard dimension and shape follow the American Society Test Method (ASTM) standard. Next, based on the size of selected product, the suitable mold based size is determined which is available on our lab. The selected and most suitable mold based is selected and do the dimensioning details with all components These task will performed with suitable tools and equipments which are ruler (mm), venire caliper and measuring tape (mm). All this dimensions is used as references to redraw in CAD software to do the modeling and analysis by CAE Moldflow Software. Next, drawing of selected sample of product and mold based as a modeling and for analysis and design details. All these drawing were draw by CAD software which is *SolidWorks*. This drawing is details with dimension as actual size as an original product. The dimension was following the current parts and also gets the reference of dog bone by ASTM standard. Next, the process continues with the material selection as sample of product material to analyze the material flow. This material is scopes to engineering plastic material which is using *Acrylonitrilebutadiencestryene* (ABS) material. All the information and data collection and parameter require for this project progress will recorded as a reference for the analysis task on PSM 2. After that, task is preparation of progress presentation and report writing chapter one, chapter two and chapter three as a complete task for PSM 1. These tasks take two week to be finish. In the presentation, the information and flow of project was explains to the panels. That is details with project objective and scope that focus in. On that particular week, preparations also include the slide presentation for the PSM 1 work progress presentation.

PSM 2: The project task for PSM 2 will continue follow as the planning. The actual work and planning for PSM 2 were present at the beginning of the semester. Project progress will be continuing with design the mold insert for dog bone tensile test specimen. The designs start with determined the suitable runner system for dog bone mold insert. The runner system will determine on the diameter and shape of cross-section that is recommended and suitable as desire shape and size of dog bone specimen. Next is determining the gating systems which are gate type, gate size and gate location. Type of gate that will choose is based on the type of injection mold which is two plate molds. Then the gate is selected for surface gate. The gate dimension will determined based on the finding of type of gate selection. The types of gate will selected and evaluate from several gate type that is possible for two plate mold. The final stage on gate mechanism is the gate location. Next, the task continue with determine the possible gate location for selected product (dog bone). This process requires the software support to determine the best gate location for the dog bone part. This process will run in several times with various locations to determine the most suitable location. The sequence of analysis that will use is flow analysis to see the material flow in the part. This process will details investigate in order to suite the applications and usage of the part (dog bone). This process will take a few weeks to done it but it need to make in fast for continue on next task. The data and result from the analysis will be as the references to continue the design of mold insert of product (dog bone). The parameters and data used are recorded in the table for proper reference and guides line. This stage is to do the modeling of the inserts how it will machine or fabricated to the mold based. This modeling can give the clear view of actual insert design like real. This modeling also can be the benchmark of improvement in complete mold constructions for two plate mold of dog bone insert. This task will performed with complete dimensioning and drawing details as final works.

This is also important to avoid the ejector pins are located at wrong placed that will cause the defect to the product during ejection process. The suitable number of ejector pins also can be determined after the solid modeling of complete design.

The data and information will record as a result achieved. This result includes the gate location, ejectors pins location, numbers of ejector pins, suitable runner size and some conclusion and discussion. This result achieve will be as a reference to future works which is fabrication as a real products.

This is how the engineer or mold insert fabricate person do to avoid the mold broke after fabrication done. This stage will observe the result that is capable for best location of gate and runner size for our sample project.

Lastly, the final report writing and prepare the presentation. This takes about one to two weeks to arrange and accomplish. A report is guided by FKM thesis format and also guidance from supervisor. All task scheduled is take around two semesters to complete. Before the complete report submission, we need to present all the results achieve from PSM 2 to the panel. The project will evaluate and have some comment for make a little bit modification for report done.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

The best way is the molds and parts designer must have a good knowledge on the basic of injection molding process. It will help them to design for manufacturer and not just a design which is very nice but it cannot be manufactured. The basic theory of plastic injection molding including the injection molding machine, injection molding process, injection molds, type of gates, plastic material, plastic parts design guideline, and machining process in molds fabrications will discuss details in this chapter.

2.2 INJECTION MOLDS

2.2.1 Types of Injection Molds

The molds are the most important component of the injection molding process. It will determine the finish size of the parts that is producing, the surface finish of the final product and dictate just how well the injection molding process will run (L.Sors and I.Balazs. 1989).

2.2.1.1 Cold Runner Injection Molds

The majority of plastic injection molds built using the cold runner type. In a cold runner mold, the plastic melt is injected into the mold through a sprue and runner where it enters each cavity in the mold. During the cooling stage of the injection molding cycle, the plastic in the cavities, sprue and runner solidify. The sprue and runner become scrap in this type of injection mold. They are then ground into small pieces or pellets and mixed with the virgin plastic for reuse. Cold runner injection molds are more economical to build than hot runner molds, however they can be less economical to run if the amount of plastic in the sprue and runner exceeds 35% of the plastic that is injected into the part cavities (L.Sors and I.Balazs. 1989).

2.2.2 Two and Three-Plate Molds

Mold for plastic injection molding can be categorized into two main categories that are two plate mold and three plate mold. Each type has a different design, function and structure. The selection types of mold depend on types of product, function and production capacity. The cost of mold fabrication also depends on the types of mold. This study will discuss details on two-plate molds.

2.2.2.1 Two-Plate Molds

This kind of mold is used for parts that are typically gated on or around their edge, with the runner in the same mold plate as the cavity. Two-plate mold can divided into 3 category that is cold runner, hot runner and conventional. Figure 2.1 shows the two-plate mold which are separated by one line called parting line into two side, core side and cavity side. Core side content ejector pin, core insert which is moving after plastic inject into the mold and the ejection bar will push the ejector pin and the ejector pin then push the part out from the mold. Cavity side is a fixed side, not moving and content feed system (sprue, runner and gate). Plastic material will injected from the machine nozzle into a sprue bush and through the runner to the cavity. Mold temperature controlled by temperature controller. That have 5 medium usually used to control the mold temperature that is chiller (3°C to 35°C), normal water (35°C to 45°C), hot water (45°C to 90°C), hot oil (90°C to 250°C) and ethylene glycol (90°C to 250°C) (A.B. Glanvill and E.N. Denton. 1985).

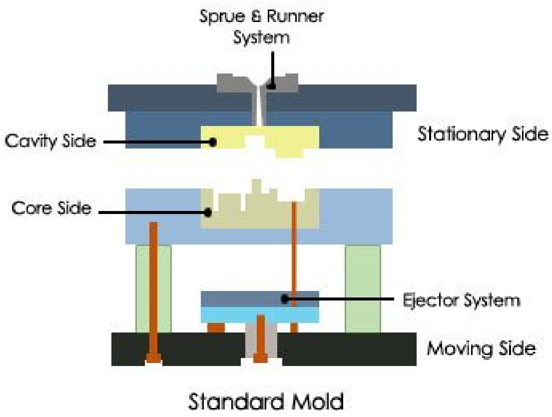


Figure 2.1: Two-Plate Mold.

Source: E. Alfredo Campo. (2006)

2.2.2.2 Advantages Of Two-Plate Molds

The advantages of the two-plate molds are (A.B. Glanvill and E.N. Denton. 1985):

- Low fabrication cost
- Simple mold/tooling design
- Easy to setting injection molding machine parameter during molding
- Easy to maintenance the mold
- Short lead time for mold fabrication
- Easy to fitting and assembly the mold
- Simple ejection system of mold

2.2.2.3 Disadvantage of Two-Plate Molds

The disadvantages of two-plate molds are (A.B. Glanvill and E.N. Denton. 1985):

- Runner and part not automatically separated (need secondary process to cut)
- The gate area can be seen clearly after cut

2.3 DESIGN OF RUNNER SYSTEM

2.3.1 Gates Characterization of The Complete Runner System

The runner system accommodates the molten plastic material coming from the barrel and guides it into the mold cavity. Its configuration, dimension and connection with the molded part affect the mold filling process and therefore largely the quality of the product. A design with a primarily based on economic view points (rapid solidification and short cycle) is mostly incompatible with quality demands especially for technical parts.

A runner system usually consists of several components. This is particularly evident in multi-cavity molds. Runner system composed of:

- Sprue
- Runners
- Gate

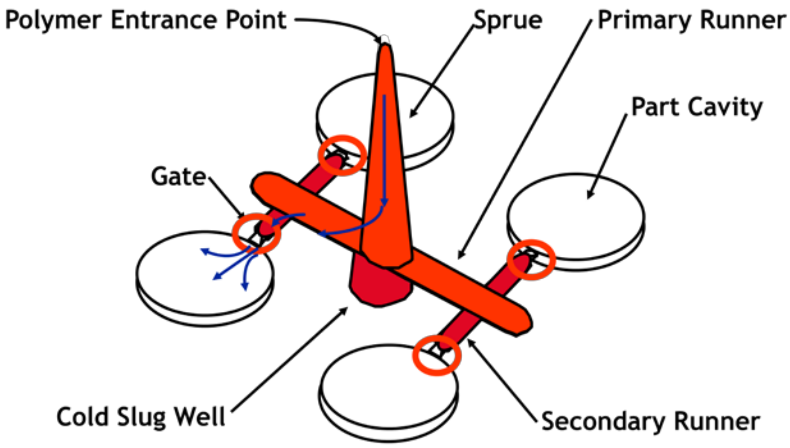


Figure 2.2: Runner Systems for Two Plate Molds

Source: Plastic Technology. 2009

The sprue bushing receives the plasticated material from the injection nozzle, which closed off the barrel and is pressed firmly against the sprue bushing. Frequently, a single cavity mold has only a sprue; the part is then said to be sprue-gate. With multi-cavity molds, the sprue bushing feeds the melt into the runners. These are connected to the cavities via the gates.

The gate is an area of narrow cross-section in which flow restricted. Its purposes are fourfold:

a) To separate the molded part easily and cleanly from the runner system
b) To hold back the cooled skin that has formed on the cold walls of the runners (avoiding flash on mold part)
c) To heat the melt through shear before it enters cavity
d) Since the cross-section of the opening can be readily altered, the runner system can be balance in such way that the melt enters each cavity at the same time and in the same condition.

2.3.2 Classification of Runner Systems

The design engineer can choose from the large number of runner systems to offer the optimum quality and economics to the user. These are:

I. Runner which remain with the mold part and have to be cut off afterwards
II. Runner which are automatically separated from the molded part and are demolded separately.
III. Runners which are automatically separated from the molded part during demolding but remain in the mold.

Table 2.1: Classifications of Gate Type

	Gating Systems
	1. Sprue Gate
	2. Edge Gate
I	3. Disk Gate
	4. Ring Gate
II	5. Tunnel Gate (submarine)
	6. Pinpoint Gate (in three-plate mold)
	7. Pinpoint Gate (with reserve sprue)
	8. Runnerless Gating
III	9. Runner for Stack Mold
	10. Insulated Runner
	11. Hot Manifold

Source: M.B. Douglas. (2008)

2.3.3 Gating Systems Designs

2.3.3.1 Design of Runner

Runners connect the sprue via the gate with the cavity. They have to distribute the material in such a way that melts in the same condition and under the same pressure fills all cavities at the same time.

The plasticated material enters the runners of a cooled mold with high velocity. Heat is rapidly removed from the material close to the wall by heat transfer, which then forms a skin. This provides a heat-insulating layer for the material flowing in the center of the channel.

This requirement in the one hand and the wish for minimal pressure loss and maximum material savings on the other, determines the optimum geometry of the runner. The dimension of the runner obviously depends on the maximum thickness of the molded part and the type of plastic being processed. The thicker the walls of the molded part, the larger the cross-section of the runner must be. As a rule, the cross-section must be roughly 1 mm larger than the molded part is thick. A large cross-section promotes the filling process of the mold because the resistance to flow is smaller than in thin runners of the same length. It pays therefore to dimension the runner according to hydraulic laws. Menges and G.Mohren. (1993)

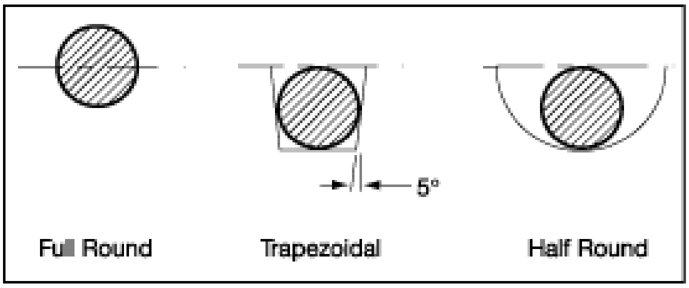


Figure 2.3: Three conventional runner profiles

Source: Menges and G.Mohren. (1993)

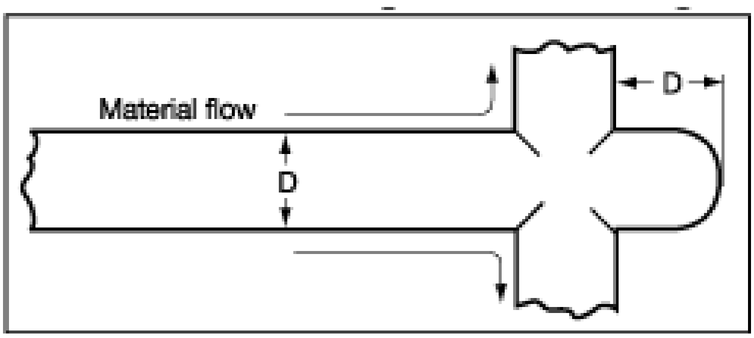


Figure 2.4: Recommended design of cold slug well.

Source: Menges and G.Mohren. (1993)

The effective size of the runner can be calculated from equation (1), (AC Technology, 1996). D is diameter of the runner in mm, W weight of the part (g) and L is the length of runner in mm. The equation is:

$$D = \frac{\left(W^{1/2} L^{1/4}\right)}{3.7} \quad \text{..................(1)}$$

But, normally practiced by mold maker to determine the effective runner is based on equation (2). D is the diameter of the runner in mm and t is the wall thickness of the part in mm. The equation is:

$$D = (2.0 \text{ to } 3.0)\,t \quad \text{...............(2)}$$

2.3.3.2 Design of Gate

Gate design is a major decision in mold construction. Gate location, size, and type will influence ease of molding, part dimensional stability, toughness, appearance and need for trimming or not. Problems relating to venting, core deflection and weld or knit lines can be prevented or solved using the proper gate and location.

The gate connects the cavity or molding with the runner. It is usually the thinnest point of the whole system. Size and location are decided by considering various requirements:

a) It should be as small as possible so that material is heated but not damage by shear
b) It must be easy to demold
c) It must permit automatic separation of the runners from the molded part, without leaving blemishes behind on the part.

The gate can be design in various configurations. Thus, one distinguishes between a pinpoint and an edge gate. In all gate types, except for the sprue gate, the gate is always the narrowest point in the gating system.

When flowing through narrow channel like a runner or gate, the material encounters a considerable resistant to flow. Part of the injection pressure is consumed and the temperature of the melt is noticeably raised. This is desirable effect because:

- The melt entering the cavity becomes more fluids and reproduces the cavity better
- The surrounding metal is heated up and the gate remains open longer for the holding pressure.
- The optimum gate size that will not cause; Thermal damage to the plastic, Too high pressure loss, has to be determined by computation or experiment during a sample run. The runners can be balance at the same times.

2.3.3.2.1 Type of Gate

Gate can be categorized into two main categories which are gate need manually trimmed and gate with automatically trimmed. Gates which needs manually trimmed, requires an operator to separate parts from runners during a secondary operation.
These types of gates include:

- Sprue gate
- Tab gate
- Edge gate
- Fan gate
- Ring or Diaphragm gate
- Flash gate

Automatically trimmed gates incorporate features in the tool to break or shear the gate. These types of gate should be used to:

- Avoid gate removal as a secondary operation
- Maintain consistent cycle times for all shots
- Minimize gate scars

These types of gate include Pin Point gate, Sub-marine gate and Hot Runner.

Direct (Sprue) gates

Direct or sprue gate (refer figure 2.5) may be used with single cavity molds where the sprue feeds material directly into the cavity. Cold slug must be providing opposite the sprue gate if possible. This type of gate is recommended for single cavity molds or for parts requiring symmetrical filling. This type of gate is suitable for thick sections because holding pressure is more effective. A short sprue is favored, enabling rapid mold filling and low-pressure losses. The disadvantage of using this type of gate is the gate mark left on the part surface after the runner (or sprue) is trimmed off. Freeze-off is controlled by the part thickness rather than determined the gate thickness. Typically, the part shrinkage near the sprue gate will be low; shrinkage in the sprue gate will be high. This results in high tensile stresses near the gate (AC Technology, 1996).

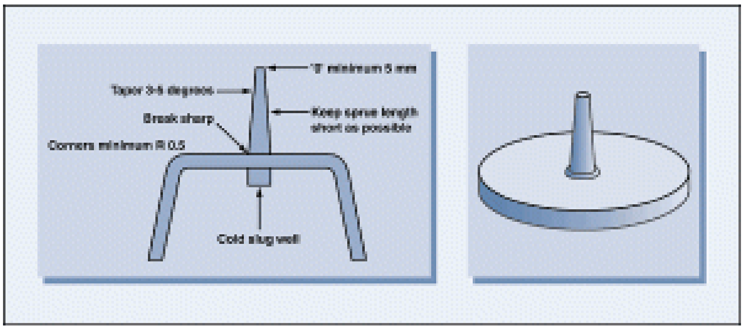


Figure 2.5: Direct (sprue) Gate

Source: Injection Molds. (2006)

Pin Point Gates

Pin gates (refer figure 2.6) actually small gates located on the parting line that keep blemishes to a minimum. They are especially popular for automatic de-gating in three plate molds because it must be ejected separately from the part in the opposite direction. The gate must be weak enough to break off without damaging the part. This type of gate is most suitable for use with thin sections. The design is particularly useful when multiple gates per part are needed to assure symmetric filling to all areas of the part (AC Technology, 1996).

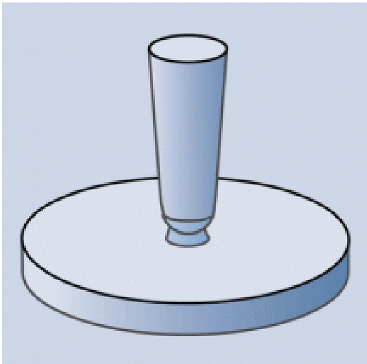


Figure 2.6: Pin Point Gate

Source: Injection Molds. (2006)

Sub-marine gates

A submarine gate (refer figure 2.7) is used in two-plate mold construction. An angled, tapered tunnel is machined from the end of the runner to the core, just below the parting line. As the parts and runners are ejected, the gate is sheared at the part. The tunnel can be located either in the moving mould half or in the fixed half. A sub-marine gate is often located into the side of an ejector pin on the non -visible side of the part when appearance is important. To de-gate, the tunnel requires a good taper and must be free to bend. A sharp angle should range between 20° to 30°, with a 60° angle from the runner (AC Technology, 1996).

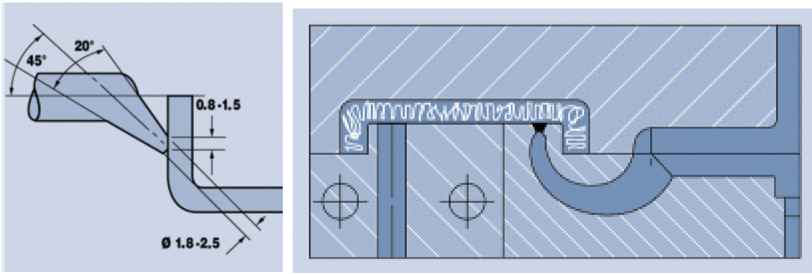


Figure 2.7: Sub Marine (tunnel) Gate

Source: Injection Molds. (2006)

Ring or Diaphragm Gate

The diaphragm or disc gates (refer figure 2.8) are suggested to used for a cylindrical parts that required good concentricity and the elimination of weld lines (for high strength). The gate land or membrane should be significantly thinner in cross-section than the runner ring or central disc. The thickness differential forces the ring or disc to fill completely before the melt fills the membrane. This type of gate requires a post mold de gating operation (AC Technology, 1996).

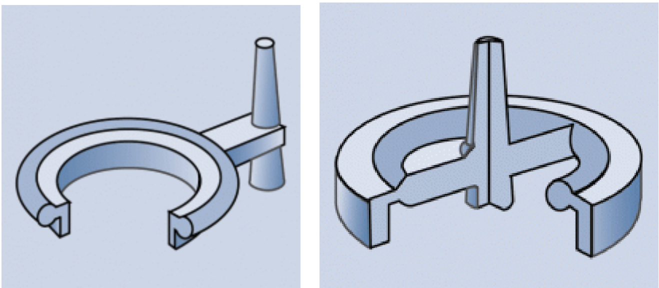


Figure 2.8: Ring (diaphragm) Gate

Source: Injection Molds. (2006)

Edge Gates

Edge gates (figure 2.9) is the most commonly used type of gate. Edge gates, both rectangular or round, must be large enough to avoid frictional burning. The edge or side gate is suitable for medium and thick sections and can be used on multi cavity two plate molds. The gate is located on the parting line and the part fills from the side, top or bottom of the molded part. The preceding figure details typical depth to width ratios for edge gate (AC Technology, 1996).

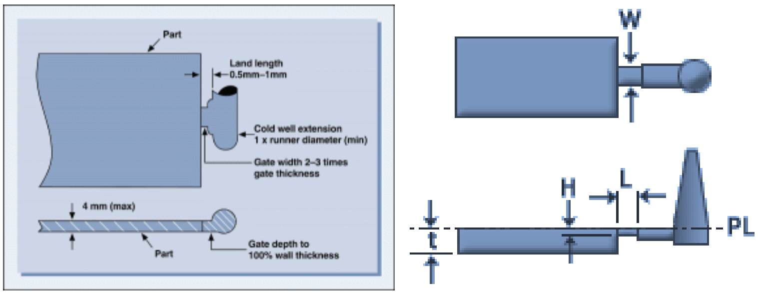


Figure 2.9: Edge Gate

Source: Injection Molds. (2006)

Fan Gates

For the flat mold sections, a modified fan gate (refer 2.10) may be used that minimizes jetting and significantly reduces the high stresses that occur during mold packing. Fan gates can be balanced so that they distribute the melt evenly across the land before the melt enters the mold cavity (AC Technology, 1996).

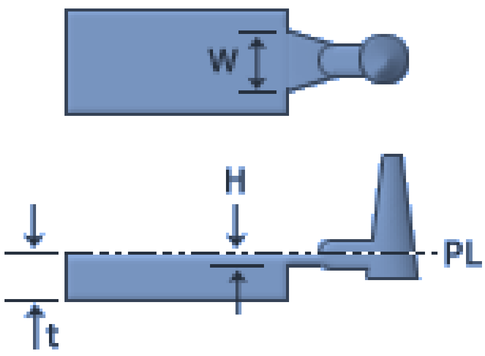


Figure 2.10: Fan Gate

Source: Injection Molds. (2006)

Tab Gates

Tab gates (refer figure 2.11) provide a uniform melt orientation, when an application requires a large volume for mold filling. The tab gates help to reduce the effect of residual stress, gate blush and jetting in the gate area. They are used where flatness is critical or in large surface areas that may have a tendency to warp (AC Technology, 1996).

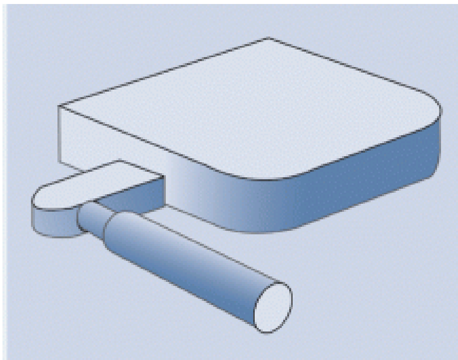


Figure 2.11: Tab Gate

Source: Injection Molds. (2006)

Flash Gates

A flash gate (figure 2.12) is a variation of a fan gate. It is used to minimize warping in flat or very large parts. Flash gates extend across a part from 0.38mm (0.015”) to 0.78mm (0.030”) deep and have a chisel taper across the gate land length (AC Technology, 1996).

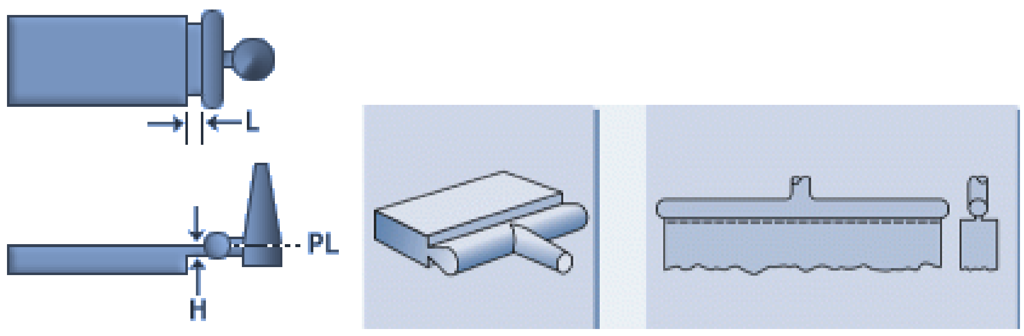


Figure 2.12: Flash Gate

Source: Injection Molds. (2006)

Table 2.2: Summary if gates type

Gates Summary	
a/ Sprue Gate	Not suitable for two cavities
b/ Pin Point Gate	This type of gate usually used in three plate mold
c/ Sub-Marine Gate	Can be consider with complex part
d/ Ring/Diaphragm Gate	This gate type is recommended for the cylindrical part. Not suitable for this part
e/ Edge/Side Gate	Most recommended because the size and wall thickness of part
f/ Fan Gate	Not suitable place to the position Will affect the unbalance flow

2.3.3.2.2 Gate Location

The gate location affects the mold filling and flow orientation. Gate location is just as important as gate size and type. Gate must be located at the proper location to ensure rapid and uniform mold filling.

Below is a general guideline to determining gate locations (Menges, G.Mohren. 1993):

- Gate should be located to feed the thickest section of the part, and then the material will flow from thick to thin sections. This will promote uniform flow and good packing.
- Optimum gate location will direct the melt flow through the length of the part with nearly equal flow distances to all its edges.
- Gate must be placed in an unstressed (nonfunctional) area of the part because the gate area tends to contain high residual stresses from the filling
- Gate must be located to direct trapped air towards the vents. Locate venting opposite the gate at the farthest flow point.
- Avoid gating locations that will cause melt fronts to converge or backfill (race track), thus entrapping air.
- Locate a gate so that it directs material towards the wall or a pin instead of into a free area. This will cause the material to disperse and continue to flow instead of jetting.
- Gate must be position to minimize weld line problems.
- To avoid multiple weld lines and venting problems, use as few gates as possible to fill the part.
- Use center-gating for round or cylindrical parts to maintain concentricity.
- Gate land lengths should be kept as short as possible, from 0.50 mm (0.020inch) to 0.75 mm (0.030inch). A too-long land length can cause poor filling, high shear and high stress in the molded part.
- Gate size must be kept as small as possible.

Because the gate area is often highly stressed, it should be located so that the product's properties and appearance are not adversely affected.

Gate location should (Menges, G.Mohren. 1993):

- Ensure a balanced flow (rapid and uniform filling) in the cavity so that certain areas of the part are not over packed
- Ensure mold fills under realistic temperatures and pressures
- Minimize weld lines as much as possible, or position them in non critical areas
- Prevent "jetting" by positioning the gate so that the material flow is smooth and uniform
- Avoid air entrapment
- Gate into the thickest section and direct material flow from thick to thin sections

The gate locations also require to follows the characteristic of part or product require. The characteristics of product that should be considered as a list below:

Appearance

Whenever possible locate gates on non-visual surfaces thus eliminating problems with residual gate vestiges after the gate has been removed.

Stress

Avoid areas exposed to high external stress (mechanical or impact). The gate area has high residual stresses and also rough surfaces left by the gate act as stress concentrators.

Pressure

Locate the gate in the thickest section to ensure adequate pressure for packing out the part. This will also help prevent sink marks and voids forming.

Orientation

Molecular orientation becomes more pronounced in thin sections, the molecules usually align themselves in the flow direction. High degrees of orientation result in parts having un-axial strength, resistance to loading only in one direction. To minimize molecular orientation the gate should be located so that as the melt enters the cavity it is diverted by an obstruction such as the cavity wall or an ejection pin.

Weld lines

Place gates to minimize the number and length of weld lines or to direct weld lines to positions that are not objectionable to the function or appearance of the part. When weld lines are unavoidable try to locate the gates close to the weld line location this should help maintain a high melt temperature that is beneficial to a strong weld line.

2.4 ACRYLONITRILE BUTADIENE STYRENE MATERIALS

According to Mat Web website state on July 2009, this material is a terpolymer of acrylonitrile, butadiene and styrene. Usual compositions are about half styrene with the balance divided between butadiene and acrylonitrile. Considerable variation is, of course, possible resulting in many different grades of acrylonitrile butadiene styrene with a wide range of features and applications. In addition, many blends with other materials such as polyvinylchloride, polycarbonates and polysulfones have been developed. Acrylonitrile butadiene styrene materials can be processed by any of the standard thermoplastic processing methods.

ABS (refer figure 2.13) is an amorphous thermoplastic blend. The recipe is 15-35% acrylnitrile, 5-30% butadiene and 40-60% styrene. Depending on the blend different properties can be achieved.

Acrylnitrile contributes with thermal and chemical resistence, and the rubberlike butadiene gives ductility and impact strength. Styrene gives the glossy surface and makes the material easily machinable and less expensive.

Generally, ABS has good impact strength also at low temperatures. It has satisfactory stiffness and dimensional stability, glossy surface and is easy to machine. If UV-stabilizators are added, ABS is suitable for outdoor applications. This characteristic of ABS material was referring at http://www.polymerprocessing.com/ on July 2009.

2.4.1 Typical Applications of ABS

The typical application for ABS material (refer figure 2.14) such as in the automotive sides (instrument and interior trim panels, glove compartment doors, wheel covers, mirror housings, etc.), refrigerators, small appliance housings and power tools applications (hair dryers, blenders, food processors, lawnmowers, etc.), telephone housings, typewriter housings, typewriter keys, and recreational vehicles such as golf carts and jet skis.

The injection molding process table for the details material conditions that provides the melts temperature, mold temperature, material injection pressure and injection speed for ABS are shows in appendix D table of the injection molding processing conditions. That is the referent usage of ABS material.

Figure 2.13: ABS material (powder).

Source: 1999-2009 Alibaba.com. July 2009.

Figure 2.14: Sample of ABS products, safety helmets.

Source: 1999-2009 Alibaba.com. July 2009.

2.4.2 Chemical and Physical Properties

ABS is produced by a combination of three monomers: acrylonitrile, butadiene, and styrene. Each of the monomers impart different properties: hardness, chemical and heat resistance from acrylonitrile; processibility, gloss, and strength from styrene; and toughness and impact resistance from butadiene. Morphologically, ABS is an amorphous material.

The polymerization of the three monomers produces a terpolymer which has two phases: a continuous phase of styrene-acrylonitrile (SAN) and a dispersed phase of polybutadiene rubber. The properties of ABS are affected by the ratios of the monomers and molecular structure of the two phases. This allows a good deal of flexibility in product design and consequently, there are hundreds of grades available in the market. Commercially available grades offer different characteristics such as medium to high impact, low to high surface gloss, and high heat distortion.

ABS offers superior processibility, appearance, low creep and excellent dimensional stability, and high impact strength.

2.5 MOLDFLOW

The plastic flow simulation (Moldflow software) is Computer Aided Engineering (CAE) tools are used to simulate the manufacture of plastic parts and the results help the engineer's correct defects on the final design, before mould tool manufacture is completed. The above factors bring a level of complexity to injection molding that makes it necessary to use Moldflow software as CAE tools to predict and solve potential problems before they occur. Additionally, the cost of tooling for injection molds can be very high and subsequent rework increases these already high costs. All these factors combine to make injection molding an ideal application for CAE simulation using Moldflow software. (P.S. Cracknell and R.W. Dyson. 1993)

Moldflow is the one of developer software solutions that enhances the design, analysis, and manufacture of injection molded plastic parts. Common consumer products that make extensive use of plastic parts are cellular telephones, personal digital assistants, pagers, automobiles, televisions, cameras, toys and personal computers. The commercial success of each of these products often relies heavily upon reducing the time to bring new products to market, reducing engineering and manufacturing costs, and improving product quality and design. The Moldflow software is useful in all aspects of the injection molded plastic parts manufacturing process, including part designers, mold designers, manufacturing engineers and machine operators. It will enable the company to speed their products to market, decrease manufacturing costs and reduce costly design and manufacturing errors by (Ref: Moldflow.com):

- assisting part designers in the selection of a plastic material
- determining the strength, rigidity and ease of manufacturing of a given part design
- predicting the amount a plastic part will shrink or warp during production
- optimizing production conditions such as machine temperatures, injection speeds, cooling times and the locations in a mold to inject the plastic
- identifying and providing optimized solutions for adverse variations during production, and

- providing features which facilitate collaboration over shared media, such as the Internet.

Mold flow software is used by more than 3,200 customers at more than 3,700 sites in over 55 countries around the world. Representative customers include Baxter International, DaimlerChrysler, DuPont, Fuji Xerox, Hewlett-Packard, Lego, Lucent Technologies, Motorola, Nokia and Samsung (Ref: Moldflow.com).

2.5.1 Type of Moldflow analysis

The mold flow consists of two types, Mold flow Plastics Insight (MPI) and Mold flow Plastics Advisor (MPA). To simulate more accurate and get the meshing analysis, usually using Mold flow Plastics Insight (MPI) as simulator to simulate the plastic. The Mold flow Plastics Insight (MPI) suite of software is the world's leading product for the in-depth simulations to validate part and mold design. Most companies around the world have chosen Mold flow’s solutions because the software allows the user to analyze CAD solid models of thin-walled parts directly, resulting in a significant decrease in model preparation time. The time savings allow the user to analyze more design iterations as well as perform more in-depth analyses. Based on the previous analysis, proven that the solutions for all types of applications Mold flow's analysis products can simulate plastics flow and packing, mold cooling, and part shrinkage and war page for thermoplastic injection molding and injection compression molding processes. The mold flow software also create solution technique based on a solid finite element volume mesh, MPI/3D allows you to perform true three dimensional flow simulations on parts that tend to be very thick and solid in nature as well as those that have extreme changes from thin to thick.

Below is some of benefit of Mold flow simulation in injection molding process. (P.S. Cracknell and R.W. Dyson. 1993)

i. Efficient process conditions
ii. Optimum cooling for cycle time savings.
iii. Optimum gate position for minimum machine size

iv. Position weld line where you want them
v. Runner balancing for minimum scrap
vi. Eliminate gas traps, sink marks & burning
vii. Minimize clamp force requirements
viii. Control fiber orientation
ix. Reduced part shrinkage
x. Gas injection simulation
xi. Reduced war page at fast cycles

CHAPTER 3

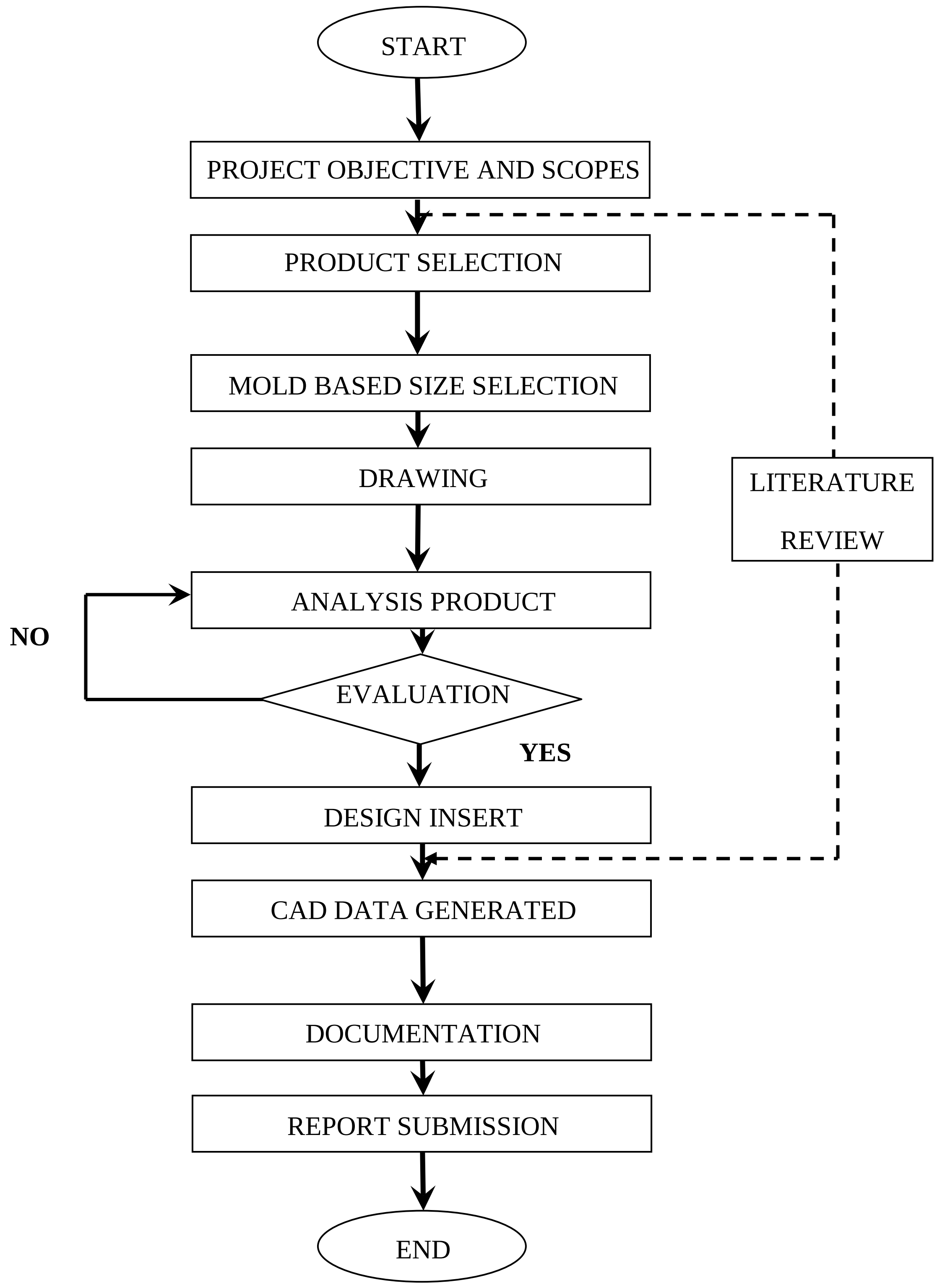
PROJECT METHODOLOGY

3.1 INTRODUCTION

In this chapter, we will discuss about the flow of project to design insert dog bone specimen of injection mold. The details of works and process will be show in the flow chart below. This flow chart will help in project progress and scheduling the task.

The methodology is as stages or steps that need to be follow and this will ensure the project done according to the planning. Methodology as an algorithm that finds a solution in the given environment of the multi-layered finite space consisting of the problems statement, project scopes and objective, literature review, product selection, dimensioning drawing, product improvement, material selection, Moldflow analysis, modeling the design and documentation.

3.2 FLOW CHART



3.3 FLOW CHART DESCRIPTIONS

3.3.1 Objective and Scope

After the title of PSM given, we need to determine what the purpose of this topic is. Then two main of problems statements are determined. First is to improve design insert of plastics product using two plate molds. Next is the improper design of gate size, gate location and runner size will caused the long time for mold testing to get the desired final appearance of the product. Cooling channel is requiring for injection molds.

From all these problems, we know the main objective of this project is to solve all the problems that were occurring. We need to analyze how to improve the material flow in two plate mold for plastic product instead the cost and appearance of the product. So, to achieve the objective of this project, we make the scopes for this project that what we will focus on this project.

There are scopes of these projects:

1. Design Insert for Two Plate Mold
2. Determine the suitable gate size, gate location, gate type, runner size for plastic product.
3. Flow analysis using Moldflow software.
4. Improve design of plastic product – Motorcycle Road Tax Holder.
5. Generate the cooling channel for two plate mold of the design.

3.3.2 Literature Review

At the beginning of this project received, supervisor was given some briefing about the title and scopes of this project. In order to have knowledge and well understanding about this project, several literature and study on website, journal, books and hand book that related was perform. The design is consider of gating system, runner design, number and cavities location. This task is performing continuously until the analysis will perform. This is to guide us and to get more knowledge and information that are related to the project.

3.3.3 Product Selection

The product selected to be analyzed in this project is a “Dog Bone Specimen” (refer figures 3.1) used as tensile test specimen to test the strength of selected material. This product is only a single part symmetrical like a dog bone shape where it is thin at the center and wide at two ends. The mold insert of this part have two cavities of the part where there is machining on one plate core (refer figures 3.3) side. For this project, this part will be designed in one mold with twin cavities, which is called family mold. This means for one shot or one cycle, the mold will produce 2 parts. We will focus on study and design with two-plate mold for this parts.

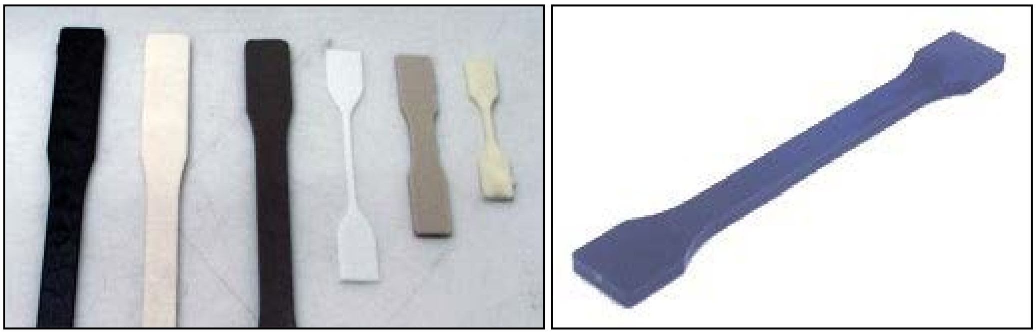


Figure 3.1: Dog Bone Specimen for Tensile Test

Source: Axel Products,Inc (2004)

3.3.4 Mold Base Size

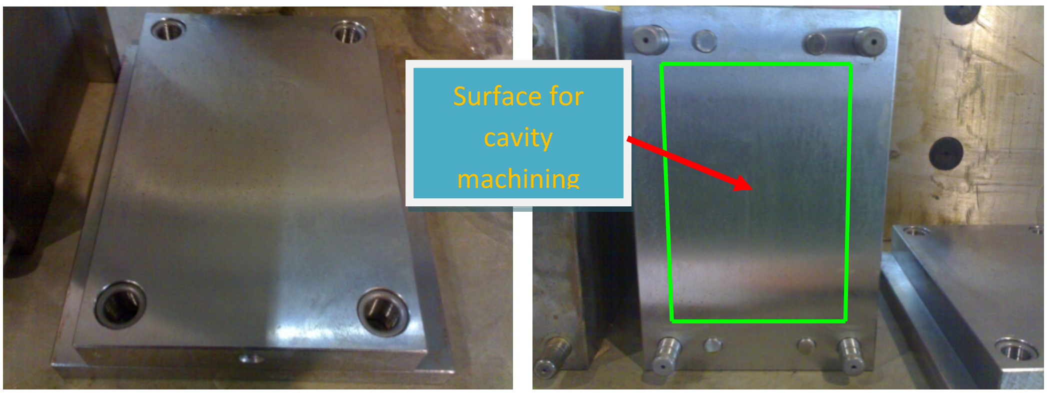
Based on overall product size and mold based that is suitable, the standard mold base recommend size is as table 3.1. So, in designing insert for this road tax holder, this mold base is suitable for project analysis as sample dimension for fabrication.

(a) Half A (b) Half B

Figure 3.2: Mold based

Table 3.1: Details Dimension for Mold Based

Plate	**Dimension:** **width × height × thickness (mm)**
Top clamping plate × 1	250 × 300 × 25
Cavity plate × 1	200 × 300 × 35
Core plate × 1	200 × 300 × 60
Side plate × 2	40 × 300 × 125
Ejector-retainer plate × 1	120 × 300 × 15
Ejector plate × 1	120 × 250 × 20
Bottom clamping plate × 1	300 × 250 × 25



(a) (b)

Figure 3.3: a) Half A core surface, b) half B cavity surface.

3.3.5 Drawing

Drawing is one of important task in this project. First, we need to draw all the components of our project need which is product and mold based. Drawing for product is drawing follow as ASTM standard which is as standard size for dog bon specimen. One size that was choose is with dimensions 163 mm in length, 19 mm in width and 4 mm in thickness. While for the mold based, it will used at the final stage as a solid modeling to see how the insert will be located and machining to the mold based. That makes easily to see the simulation before we do the fabrications. These drawings perform using CAD software which is SolidWorks that make easily view and modeling in 3D view. This software also related the analysis software require where that need IGES files where the SolidWorks can do.

Figure 3.4: SolidWorks 2006 software

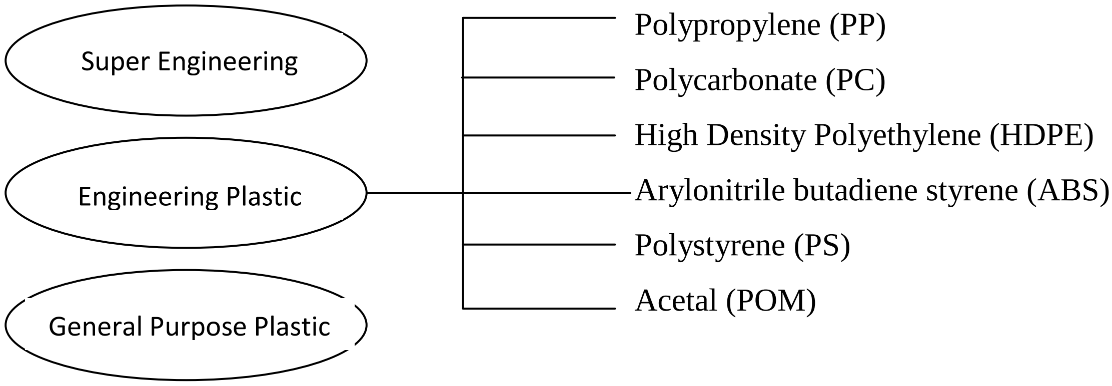
3.3.5.1 Technical Drawing

The details drawing for mold based components and the dog bone specimen can be refers to appendix B1 until B12. Drawings have isometric and orthographic view complete with dimensions.

3.3.6 Material Selection

For the sample of material that will used to analyze the material flow for dog bone specimen, the engineering material was selected with ABS material which is suitable and capable for tensile rest material specimen.

Below is the position of ABS material among the major resin.



3.3.7 Moldflow Product Analyses

The complete drawing model for product (dog bone) will analyze by using CAE software which is *Moldflow*. This process is to perform the analyses that are needed and requires. This software is available in two capabilities; a) Moldflow Plastic Adviser (MPA) and, b) Moldflow Plastic Insight (MPI). But, these analyses will perform by Moldflow Plastic Insight (MPI). The result of the analysis will used as references to design the insert of product.

The analysis for this project consists of:

1. Gate location analysis.
2. Material flow analysis.

These analyses require several information about the material usage, material manufacture, material melt temperature and recommended mold temperature for selected material.

The trade name and manufacture of material that are available in Moldflow software is shows in table of selected material with their manufacturer (refer to appendix D1). Type of material determine from where manufacture it is and the specific name for that material in the software. The analysis parameter also requests the melt temperature of material that we have defined. Then table at appendix D2 shows the recommended melting temperature for several materials for injection molding process. The mold temperature state as constants parameters like melting temperature and it temperature is recommended as appendix D3.

3.3.7.1 Gate Location Analysis

The steps of and details process for gate location analysis as the instructions below:

1. Drawing model (parts) with SolidWorks and save as **IGE**S files.(Refer to appendix E1 and E2)
2. Create a project on MPI software. (Refer to appendix E3)
3. Import model to analysis window which is selecting the PART with IGES files then select the type of mesh for **FUSION**. (Refer to appendix E4 and E5)
4. Create meshing with **4mm** node. (Refer appendix E6 and E7)
5. Then, select **Cavity Duplication Wizard** under modeling task bar menu. Select the **Row** option with two (**2**) cavities and set the distance with **50mm** in the part distance in **Row column**. (Refer appendix E8 and E9)
6. Next, select the **Gate Location Analysis** under the analysis menu with analysis sequence. Double click on analysis item on left study task bar for gate location analysis. (Refer appendix E10)
7. The result for gate location analysis will appear in result window. (Refer appendix E11)
8. Analysis the best gate location
   From result in step 7, then determine the best location of the with the flow analysis of 21 points difference gate location as shows in figure below. (Refer appendix E12).

   The analysis of material flow will focus on orientation skin effect to the part (dog bone specimen)

   a. Select flow analysis sequence. (Refer appendix E13).
   b. Set first gate location. (Refer appendix E14).
   c. Duplicate The Model To Two With **50mm** In Distance. (Refer appendix E15).

d. Set the runner with select the **Runner System wizard** under modeling task menu. (Refer appendix E16).

e. Set up the runner with **0mm** in **z axis**, **3mm** orifice diameter with **3degree**, set **50 mm in length**, set **8mm** full round runner diameter, **3mm** diameter gate with **15 degree** of angle and set **2mm** length of gate. (Refer appendix E17, E18, E19 and E20).

f. Set The Melt and Mold Temp with Automatic Cool Times under process setting on study task menu bar. (Refer appendix E21).

g. Set the material with ABS by selecting the name of manufacturer with their trade name which is **Tayolac 100**. (Refer appendix E22).

h. Run The Analysis. (Refer appendix E23).

9. Repeat steps **8b** until **8h** for next **20** different location of gate. Record all the results achieve in the table systems.

3.3.8 Design Insert

Finally, after the product finish analyzed, the insert can be design as require follow the result appear in the analysis stages. This design wills easily modeling in 3D view before we fabricated the real insert. This stage of design consists of several related elements and parameters.

Next, for twin cavities, the cavities a layout is require. Figure 3.5 below show the details measurements of cavities layouts on mold based. This layout will used to define the distance between two cavities in the Moldflow.

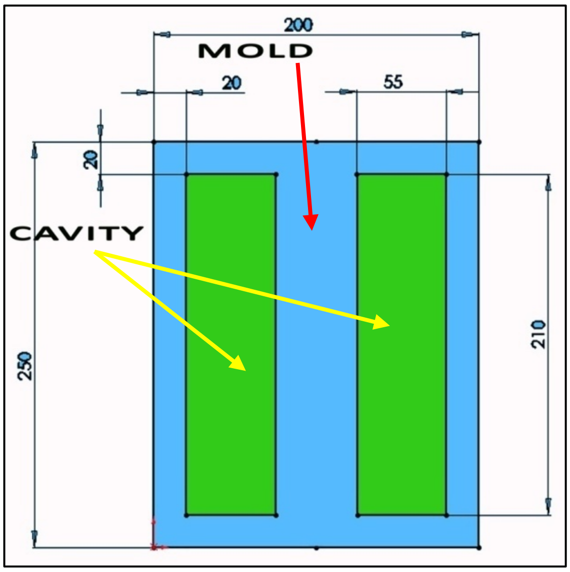


Figure 3.5: Cavities layout on mold based

3.3.8.1 Gate Size

Gate size will be calculated from the gate selection with their own calculation. The gate size also depends to the product that will produce such as thickness, the wall thickness and the shape of parts.

Gate type will evaluate by several types of gate that is possible as figure common gates type as shown at appendix E1.

3.3.8.2 Classification of Runner System

The runner systems designs based on the classification of runner systems that is stated in chapter 2 (page 12, 13).

3.3.8.3 Runner Size.

Design for runner systems was select the equation [2] at chapter 2 (page 15) where the suitable diameter size for runner is recommended.

3.3.8.4 Runner Cross Section

From table runner cross-section characteristics (refer to appendix F1), three different runner cross-section is determine. There has there conventional runner cross-section that normally used in two-plate mold (figure 2.3 in chapter 2). The evaluation is based on the advantages with fewer defects to product.

3.3.9 CAD Data Generated

As a guideline and future works reference. This modeling is valuable to generated code by using CAM software which is Master Cam. This CAD data is show as real product or design. It can easily measure any ware if needed to take the dimensions.

3.3.10 Result

The data and information will record as a result achieved that is shows in chapter 4 This result includes the product quality, process parameter injection molding machine for this mold, and some conclusion and discussion. This result achieves from new design and will compare with the current design.

3.3.11 Documentation and Report Submission

The documentation is the final report where consist of chapter one till seven. Those chapters content include Introduction, Literature review, Methodology, Result and Discussion and Conclusion. This report will be submitted to supervisor as final check before submitted to the faculty. This report will submit in thesis format of degree students with hard cover bindings as require color and course.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 INTRODUCTION

This topic will discuss how the insert of two-plate molds will be designed. Mold design is start with observe and details their components and parameters requires. We start with finalize the type of gate to be used, the number of cavities of the part, runner layout, insert size, mold based size, gate size and the gate location. Then from the material flow analysis, we will know the orientation skin for product after injection where that is suppose to fulfill the characteristic of product. Below is the information for insert to be design in this project.

- Type of Mold : Two Plate Mold
- Number of Cavity : Two Cavity
- Distance between Parts : 50mm
- Ejector system : Require

4.2 RUNNER MECHANISM

Consider the important of runner function is where to reduce the pressure when plastic material flows into the mold, the suitable runner size were determined.

From the equation [2] in chapter 2, the diameter of runner is calculated. From the part referent, the wall thickness, t =4.0mm. By using equation (2), D = 4.0 to 6.0mm. So, based on equation (2), the diameter of the runner should be between

4.0mm till 6.0mm. So, for this design, the size was decide at the middle of the range which is 5.0mm diameter.

So, for this design, the round cross-section of runner with diameter, D = 5.0mm (refer figure 4.1) was selected based on evaluation of table 4.1 below.

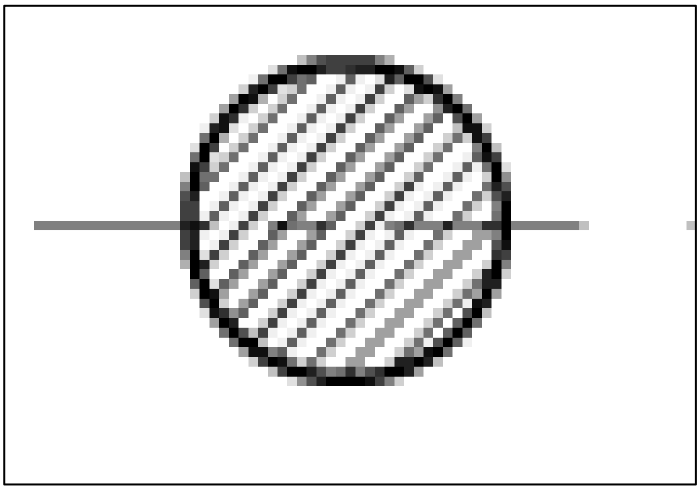


Figure 4.1: Full round runner cross-section.

The description for three type of runner cross-section was listed on table of runner cross-section description at appendix F. So, from the table the most suitable and commonly used is full round cross-section of runner. But, it is make some difficulties during machining while to align both side of half A and half B plate without miss place.

4.3 GATE TYPE

Table 4.2 shows the various types of gates. These types of gates will be select based on the part to be molded which is dog bone specimen with two plate mold type of design.

Table 4.1: Type of gate to be design for this project

Gates	Descriptions
a/ Sprue Gate	Not suitable for two cavities
b/ Pin Point Gate	This type of gate usually used in three plate mold
c/ Sub-Marine Gate	Can be consider with complex part
d/ Ring/Diaphragm Gate	This gate type is recommended for the cylindrical part. Not suitable for this part
e/ Edge/Side Gate	Most recommended because the size and wall thickness of part
f/ Fan Gate	Not suitable place to the position Will affect the unbalance flow

From the table above, the **Edge Gate** is the best selection for this design.

4.4 GATE SIZE

From the selection of gate type, we know the gate that to be used for this design is Edge Gate. The details dimension that to be consider for this type of gate size design is shown in figure 4.2 below.

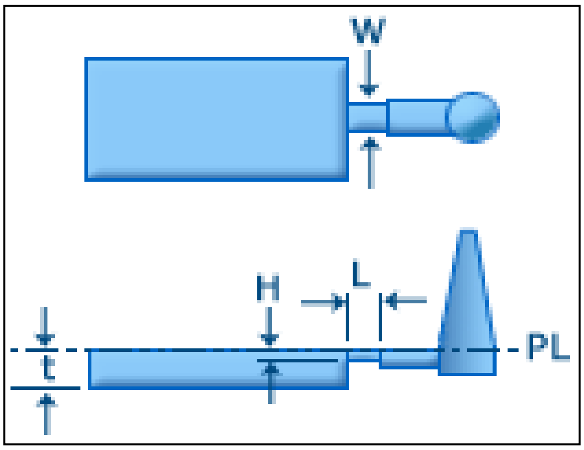


Figure 4.2: Edge Gate

An edge gate is located on the parting line of the mold and typically fills the part from the side, top, or bottom.

Dimensions

The typical gate size is 6% to 75% of the part thickness (t) (or 0.4 to 6.4mm thick (H)) and 1.6 to 12.7mm wide (W). The gate land should be no more than 1.0mm in length (L), with 0.5mm being the optimum.

Based on our product thickness which is dog bone specimen, dimension of gate size is concluding as table 4.3 below.

Table 4.2: Details Gate Dimensions

Gate	t	H	L	W
Dimensions (mm)	4	0.5	1.0	4

4.5 GATE LOCATION

From the analysis on chapter 3, we take only nine (9) points of difference gate location which are at locations 1, 4, 7, 9, 11, 13, 15, 18 and 21 (figure 4.3) that to be evaluated their orientation skin. The result of analysis as shows below (for gate location and their orientation skin appear):

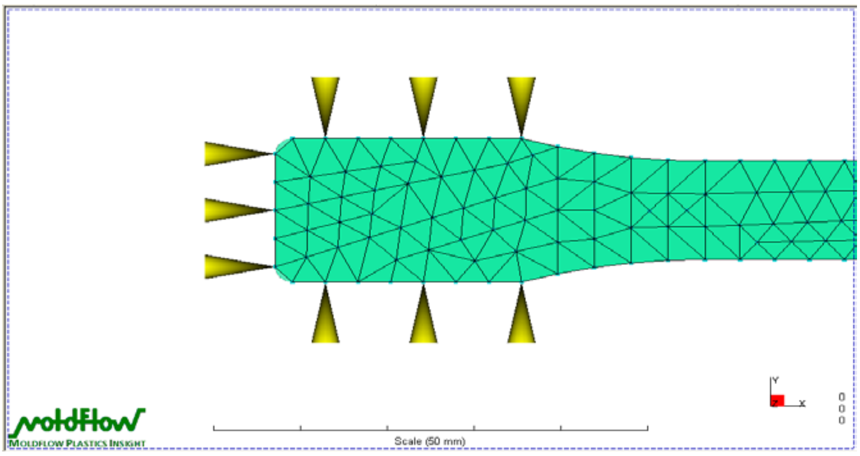


Figure 4.3: Selected gate location

First is the result at the location one (1) (figure 4.4) where the beginning location. Then the result was achieved as figure 4.5 for location of gate at point 1. The results for locations of gate at point 4, 7, 9, 11, 13, 15, 18 and 21 shows in appendix G with the orientation skins appear.

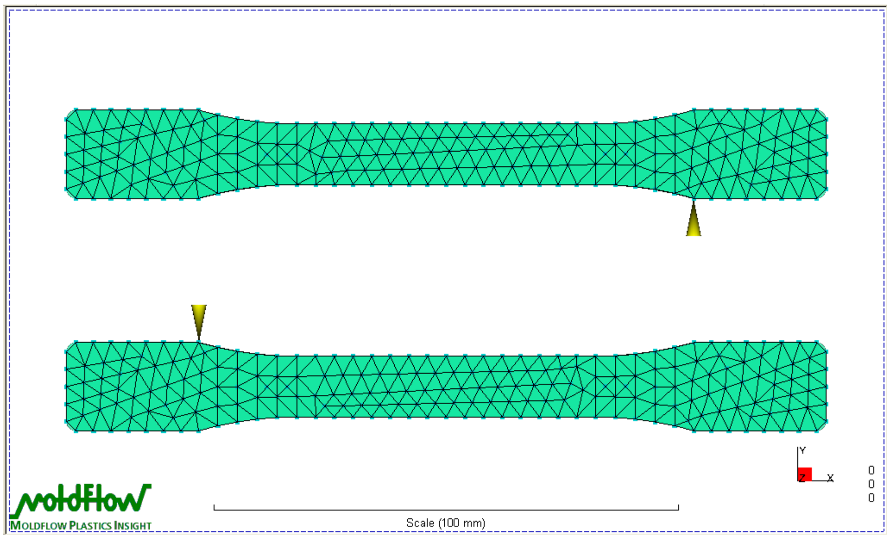


Figure 4.4: Position of gate at location 1

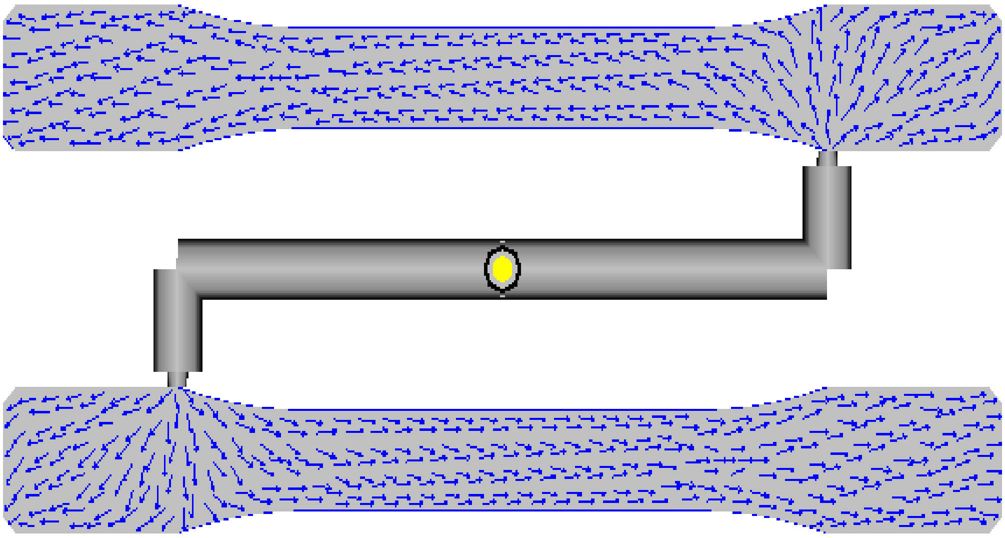


Figure 4.5: Result of orientation skin at location of gate 1

Discussion 1:

The Orientation at skin result plot is generated from a Fusion flow analysis, and provides a good indication of how molecules will be oriented on the outside, showing the average principal alignment direction for the whole element.

The skin orientation for each triangular element aligns with the velocity vectors when the melt front is first in contact with the element. This gives the most probable molecular orientation on the skin surfaces of a part.

Skin orientation is useful for estimating the mechanical properties of a part. For example, the impact strength is typically much higher in the direction of skin orientation. The tensile strength is also higher in the direction of skin orientation, because the material on the surface is aligned in that direction. Skin orientation generally represents the direction of strength. For plastic parts that must withstand high impact or force, the gate location is designed that to give a skin orientation in the direction of the impact or force.

Then, from the observation of 9 difference location of gate, we find that the best gate location is located at point 11 which is 44mm from the original point (point 1). The orientation skin appears is fulfill the requirement for specification of dog bone specimen which is to determine the strength of material. Below is summary of result:

Table 4.3: Summary of gating system

Summary result for gating system	
Type of Gate	Edge Gate
Gate Size	4mm(t), 0.5mm(H), 1.0mm(L) and 4mm(W)
Gate Location	At point 11
Runner Diameter	5.0mm

4.6 COMPLETE DESIGN OF TWO PLATE MOLD INSERT

The figure 4.6 show the complete set of two plate mold with the dog bone specimen insert. The details of parts complete design can be refer to appendix H.

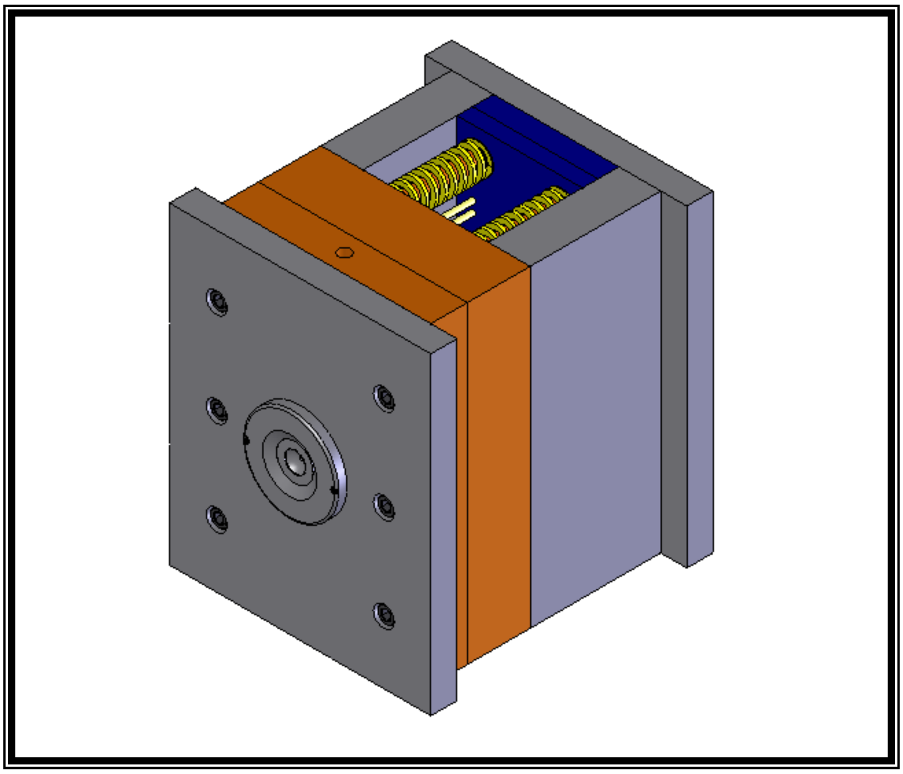


Figure 4.6: Complete Set of Two-Plate Mold

4.7 DATA COLECTIONS

From the analysis achieved, the data that was achieved was recorded. The data was divided into three elements which are the throughput in cm^3 with the position of gate (table 4.10) of material flow analysis and the graph was plotted.

Table: 4.4: Volume of Throughput with their locations

GATE AT LOCATION	[cm^3] THROUGHPUT
1	34.48
4	34.6
7	35.87
9	38.82
11	39.65
13	40.48
15	40.74
18	42.14
21	46.12

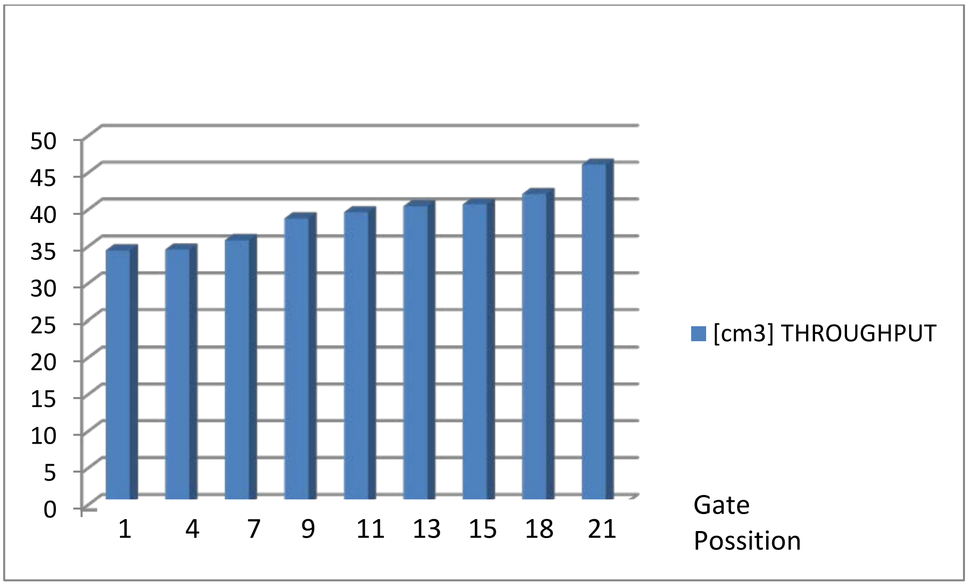


Figure 4.7: Graph of Throughput versus gate Locations

Discussion 2:

From the graph, the observation find the volume of throughput (sprue, runner, gate) was increase for every change of gate locations. The graph show, when the distance of gate location increase, the volume of throughput will increase. That is show the volume throughput increase because the runner length will increase when the distance of the gate location change. The nearest locations give the small volume while the highest distance will give the high volume of throughput. So, the best and suitable choice for gate locations is for the lowest volume of throughput. But, it is needed to consider the skin orientation of the material flow is first priority. From the result of gate location achieved, we know that the location that choosing is at point 11. So, in the graph of throughput versus position of gate show the volume of throughput for this location is at average. That is possible and can be proceed for the design and not need to change it.

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

5.1 INTRODUCTION

The final chapter is the conclusion and recommendation for this project. This chapter will conclude the project and propose the future recommendation to continue on analysis flow and parameter in designing insert for two plate mold. This project proposes to be the references for insert fabrication in the future.

5.2 CONCLUSION

The two plate mold insert for the selected part have been designed and the material flows for 21 difference gate locations have been analyzed. This chapter will conclude the project and propose future recommendations to continue on the analysis of material flow in injection molding process.

The best gate location (refer figure 5.1) for dog bone specimen insert was determined and it was fulfill the objective for this project. This location was proved in chapter 4 where show how the different orientation of skin for part with different gate position. Then, the best position is at point 11 that was stated in chapter 4.

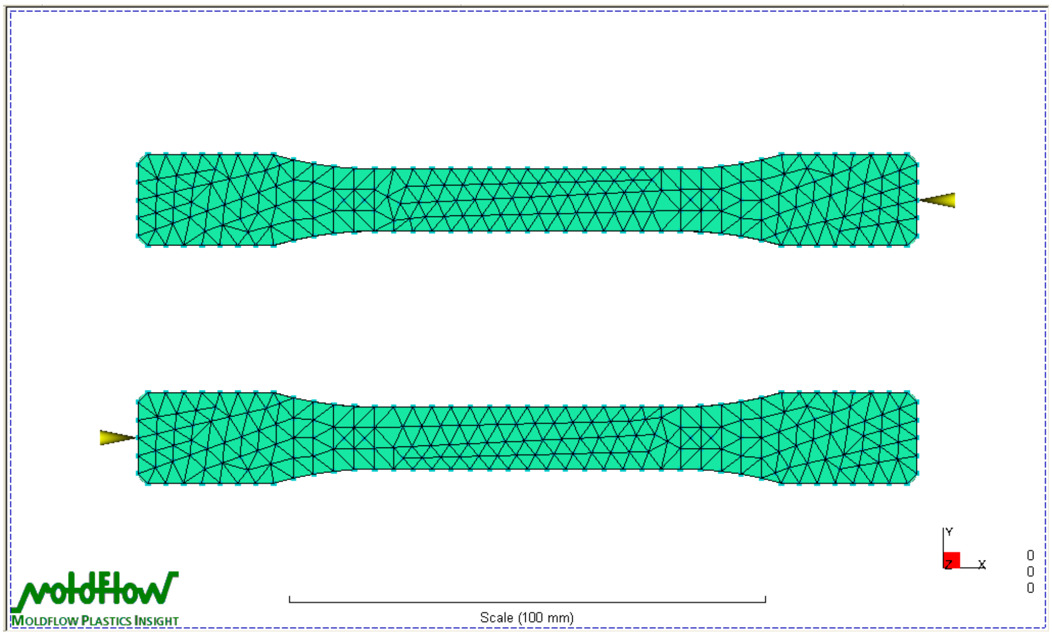


Figure 5.1: The best gate location for dog bone specimen

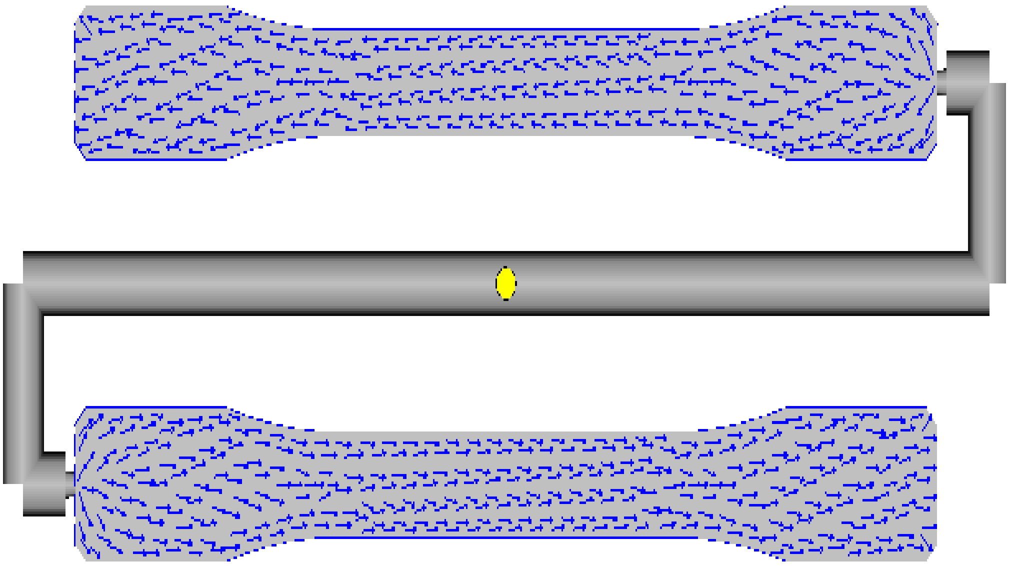


Figure 5.2: Result of orientation skin for location determine (point 11)

Finally, the two plate mold insert for dog bone specimen was design with details design, calculation and selection of their mechanism for design. This design complete with the 3D modeling and details drawing. This design also can be reference for future works which is to fabricate the mold insert. Then, the actual test run can be run to compare the simulation result with the actual process.

5.3 RECOMMENDATION

From this project, the project can be continue to the next semester where the task is proceed for the mold insert fabrication based on result of this project. The fabrication will see details how the differences and precise of simulation analysis using Moldflow software with the actual run.

Next, the recommendation goes to the faculty which is to upgrade the software that already used in lab from MPA to MPI. This is because, we can learn mold details on MPI software for material flow analysis and for mold inserts fabrication.

Lastly, we recommend the analysis or scope for the next project is to analysis the family insert with the runner balance. That is more interesting where we can reduce the fabrication cost for different size of specimen with fabricate in single mold.

REFFERENCES

This study is prepared based on the following references;

A. Brent Strong. Plastics Material And Proceesing-Third Edition. Pub. Pearsan Prentice Hall 2006.

Beaumont, J. P., Nagel, R., and Sherman, R. Successful Injection Molding, Cincinnati: Hanser Pub. Munich 2002

Defect, Injection moulding ,www.huntsman.com/pu/media/injection_molding110705.pdf / Mac 2009

Elsevier B.V. 2006 The use of Taguchi method in the design of plastic injection mould for reducing warpage.

Gastro, Injection Molds (130 Proven Design)-Third Edition. Hanser Publishers, Munich 2002

Herbert Rees & Bruce Catoen, Selecting Injection Molds, Carl Hanser Verlag, 2006.

http://tjxyhsl.en.alibaba.com/ABS_Acrylonitrile_Butadiene_Styrene / July, 2009.
http://www.plasticstech.info/processes/molding/injection-molding/process.htm/June 2009

Injection Molding Cycle & Process,PLASTICSTECH,

Injection molding history, www.plasticmoulding.ca/history.htm / Feb 2009.

John P.Beaumont, Runner and Gating Design Handbook, Hanser publisher, Munich, 2004.

K. M. Au & K. M. Yu, A scaffolding architecture for conformal cooling design in rapid plastic injection moulding, Received: 4 August 2005 / Accepted: 25 March 2006 / Published online: 8 June 2006, Springer-Verlag London Limited 2006, www.springerling.com./ Mac 2009.

Kalpakjian, Serope, and Schmid Steven R,Manufacturing Engineering and Technology. 5th Edition. Upper Saddle River, NJ: Prentice Hall. 2002

Machine Description, Arsburg 520 C 2000 – 800, www.equipmatching.com/used_ equipment.php/ Mei 2009.

Menges, Michaeli & Mohren, How to Make Injec tion Molds-Third Edition. Hanser Publishers, Munich 2001

Mold Flow catalogue, www.moldflow.com/stp/english/products/autodesk_moldflow_insight.php / June 2009.

MoldFlow plastic insight (MPI) ,www.mem.odu.edu/faculty_staff/hou/section2.pdf / June 2009.

Moldflow simulation, Imtech technology, www.imtechdesign.com/ July 2009.

Robert A. Malloy, Plastic Part Design for Injection Molding- An Introduction, Hanser Publishers, Munich 1994

APPENDIX A

FINAL YEAR PROJECT GANT CHART

GANT CHART / SCHEDULE FOR PSM I

NO.	AKTIVITIES/ TASK	WEEKS															
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
1	PROJECT TITLE RECIEVE																
2	BRIEFING AND DISCUSSION ABOUT THE PROJECT OVERALL WITH SV																
3	FIND INFORMATIONS AND DATA COLLECTIONS FROM SCOPES OF THE PROJECT THROUGH JOURNAL, WEB SITES, BOOKS AND OUTSIDE SOURCE																
4	PRODUCT AND MOLD BASED SELECTION (SIZE)																
5	INITIAL DESIGN AND DRAWING																
6	INITIAL DRAFT SUBMISSION																
7	PRESENTATION PREPARATION																
8	REPORT CORRECTION AND FINAL DRAFT SUBMISSION (PSMI)																

PLANNING ACTUAL PROGRESS

GANT CHART / SCHEDULE FOR PSM II

NO.	AKTIVITIES/ TASK	WEEKS															
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
1	CURRENT PROGRESS DISCUSS TO CONTINUE PSM II																
2	LEARN AND STUDY OF MPI SOFTWARE																
3	RUN THE ANALYSIS																
4	OBSERVATION AND OBTAIN THE RESULT																
5	INITIAL DRAFT SUBMISSION TO SV																
6	PRESENTATION PREPARATION																
7	FYP II PRESENTATION																
8	REPORT CORRECTION AND FINAL DRAFT SUBMISSION																

PLANNING ACTUAL PROGRESS

APPENDIX B

TECHNICAL DRAWING

APPENDIX C

Table C 1: The Injection Molding Processing Conditions

Drying	ABS grades are hygroscopic and drying is required prior to processing. Suggested drying conditions are 80 - 90 C (176 - 195 F) for a minimum of 2 hours. The material moisture content should be less than 0.1%
Melt Temperature	200 - 280 C (392 - 536 F); Aim: 230 C (446 F)
Mold Temperature	25 - 80 C (77 - 176 F). (Mold temperatures control the gloss properties; lower mold temperatures produce lower gloss levels)
Material Injection Pressure	50 - 100 MPa
Injection Speed	Moderate – High

Source: Moldflow Plastic Insight. (2005)

APPENDIX D

Table D 1: Table selected material with their manufacturer.

Material	**Trade name**	**Manufacturer**
Acrylonitrile Butadiene Styrene (ABS)	Toyolac 100	Toray industry incorporation
Polycarbonate (PC)	Panlite L-1225	Teijin chemicals
Polyethylene (PE)	Dowlex 2517	Dow chemical (USA)
Polypropylene (PP)	Atofina polypropylene	Atofina petrochemical
Polystyrene (PS)	Austrex 103	Polystyrene Australia
Polyethylene terephthalate (PET)	Arnite AV 2370	DSM engineering plastics

Table D 2: Table suggested melt temperature for selected plastics material.

Plastic material	**Temperature (^{o} C)**
Acrylonitrile Butadiene Styrene (ABS)	216
Polycarbonate (PC)	288
Polyethylene (PE)	204
Polypropylene (PP)	177
Polystyrene (PS)	199
Polyethylene terephthalate (PET)	232

Table D 3: Table suggested mold temperature for selected plastics material.

Plastic material	**Temperature (^{o} C)**
Acrylonitrile Butadiene Styrene (ABS)	85
Polycarbonate (PC)	104
Polyethylene (PE)	43
Polypropylene (PP)	49
Polysulfone (PS)	82
Polyethylene terephthalate (PET)	99

Source: (C.E. Alfredo. 2006)

APPENDIX E

ANALYSIS PROCEDURE/STEPS

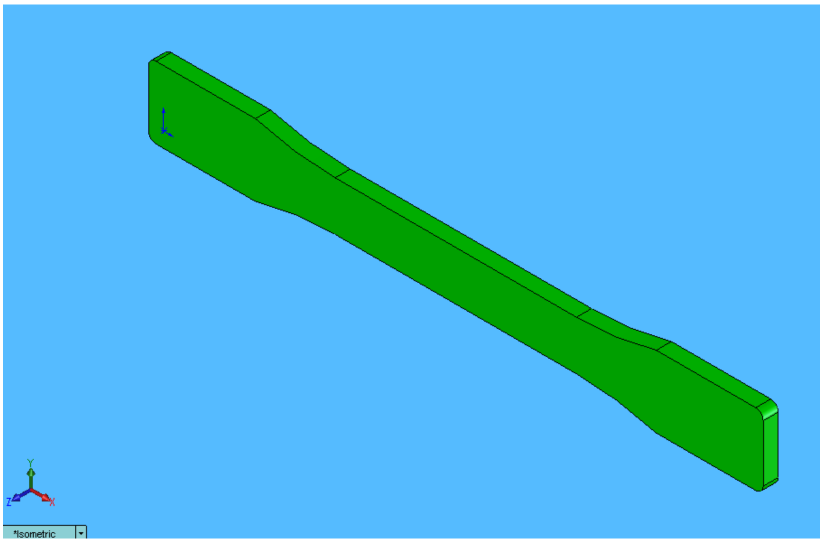


Figure E 1

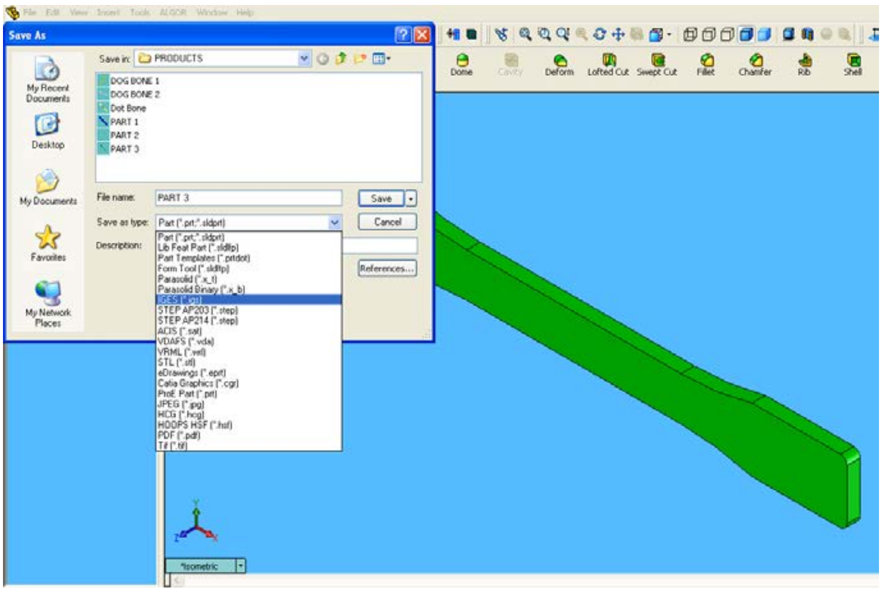


Figure E 2

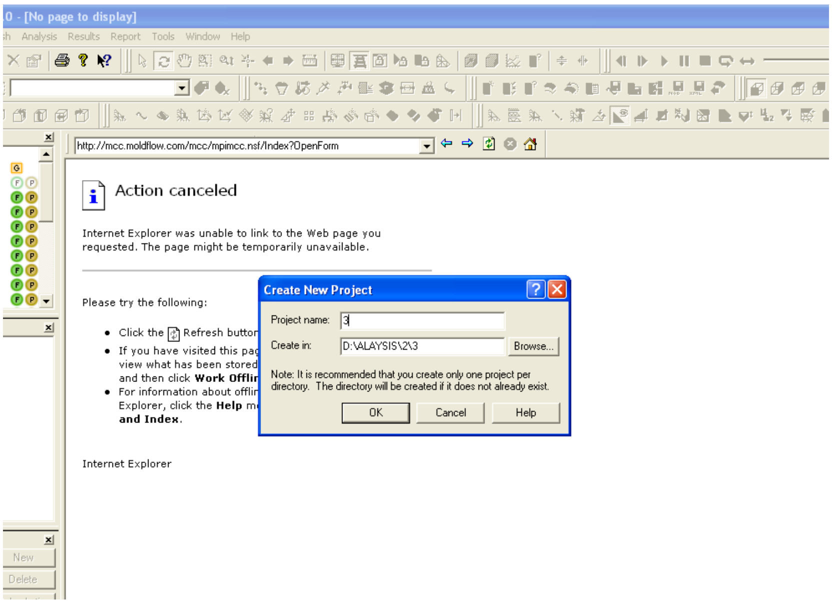


Figure E 3

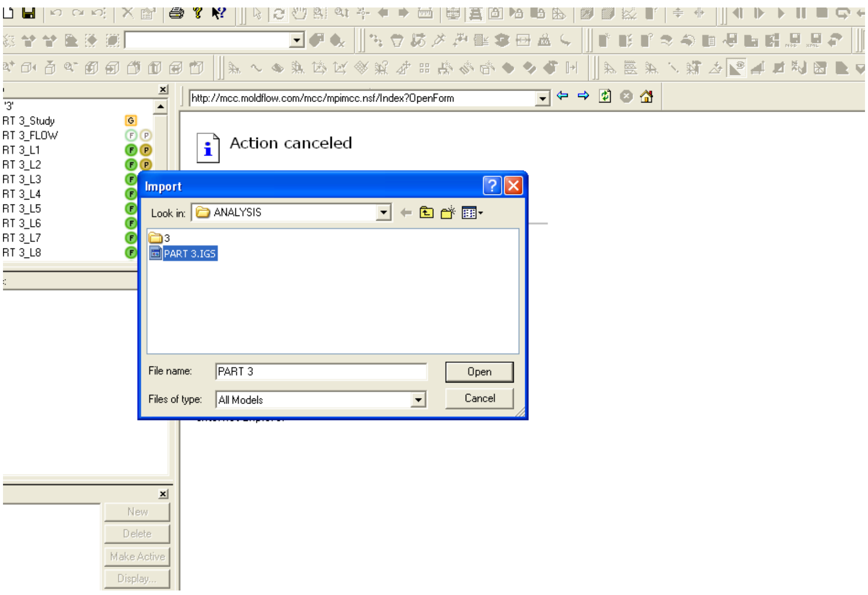


Figure E 4

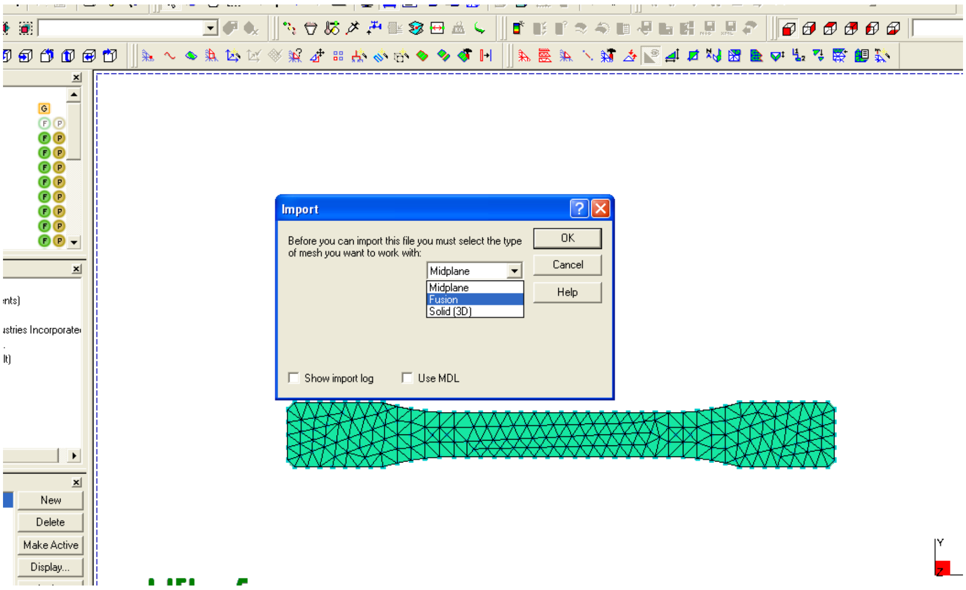


Figure E 5

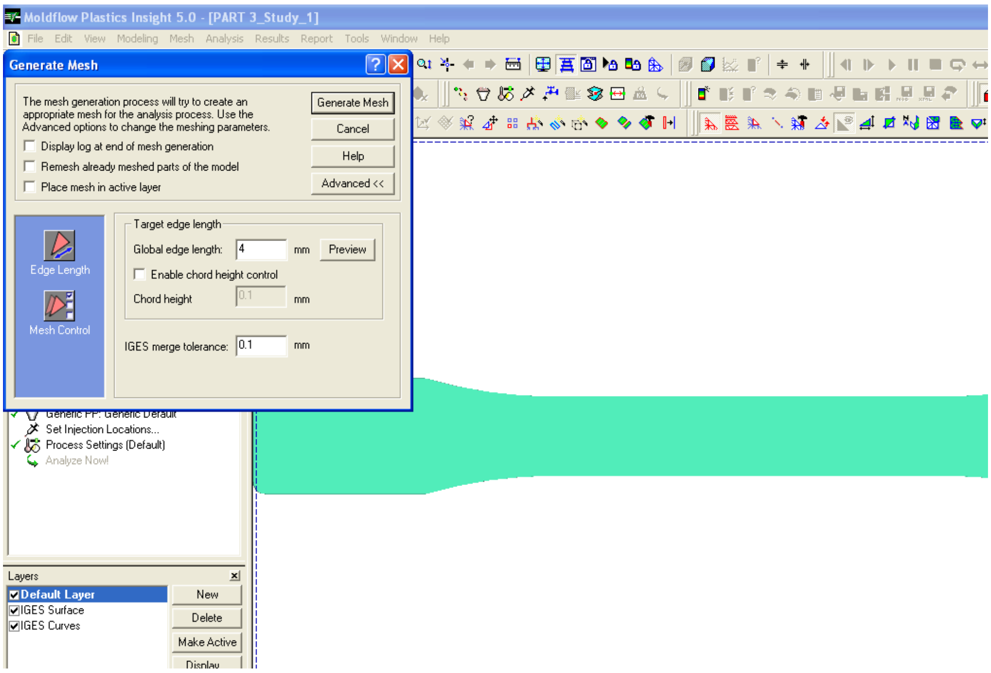


Figure E 6

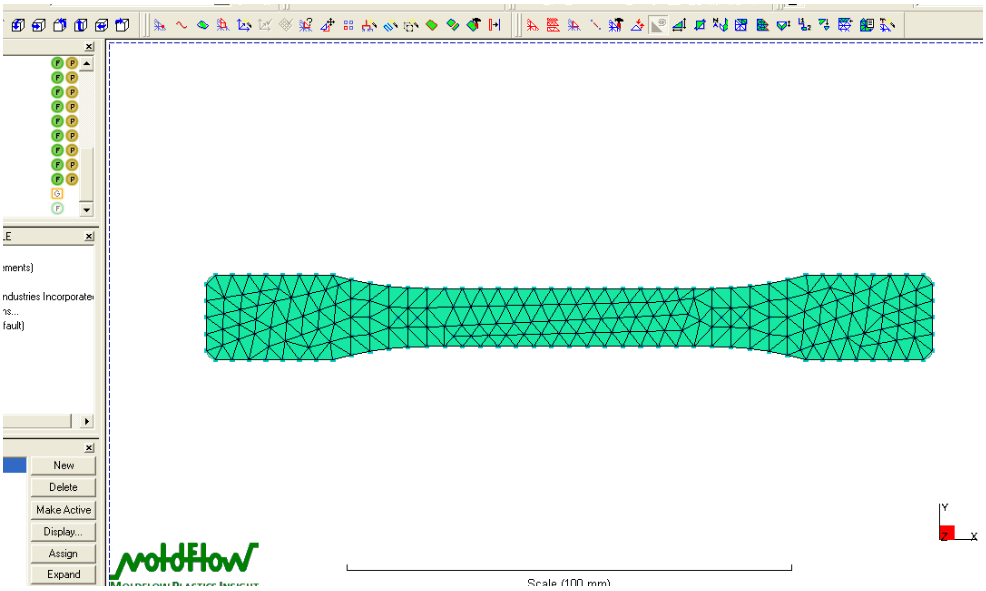


Figure E 7

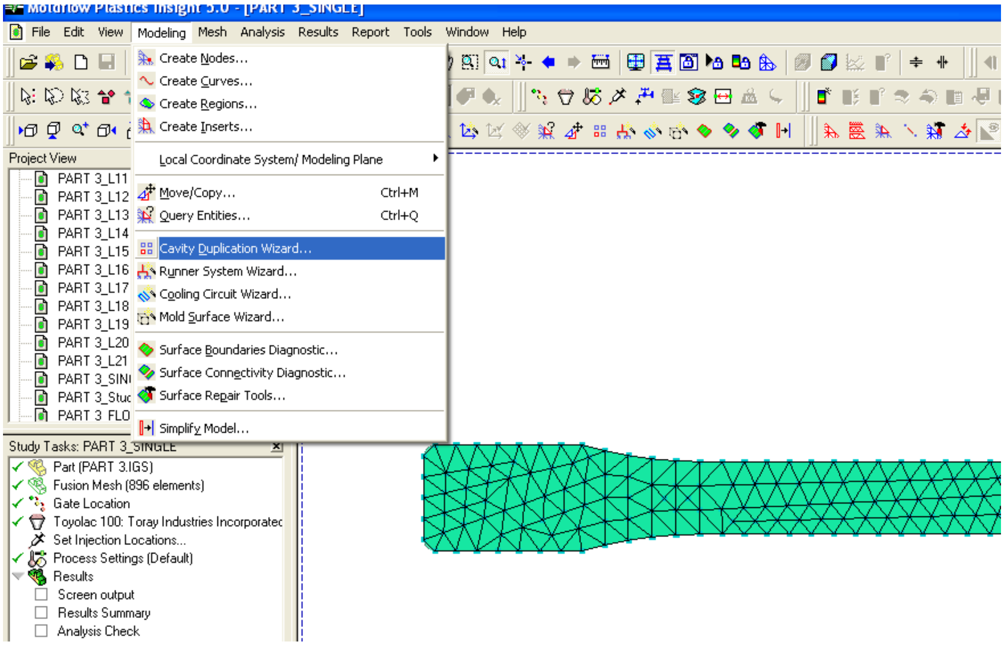


Figure E 8

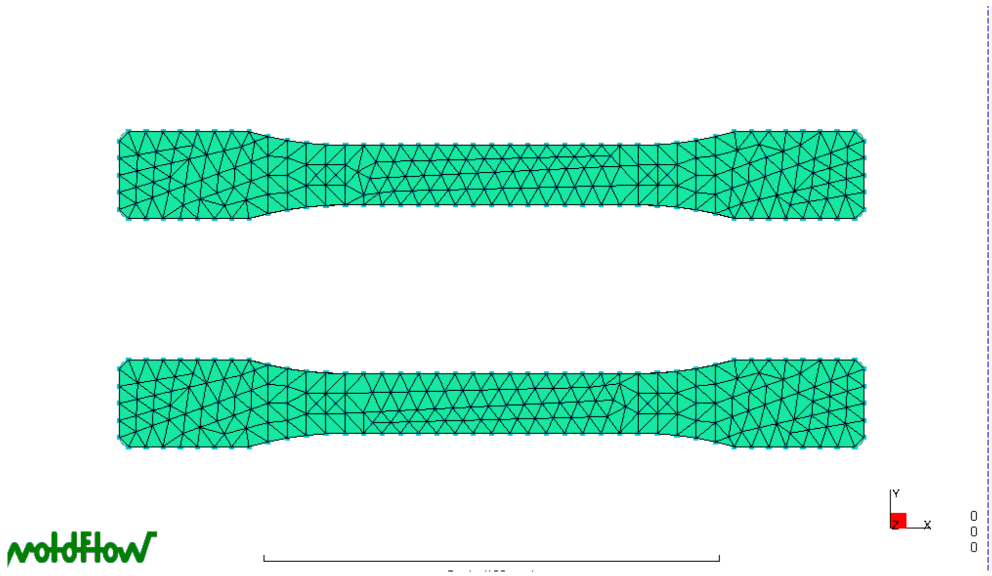


Figure E 9

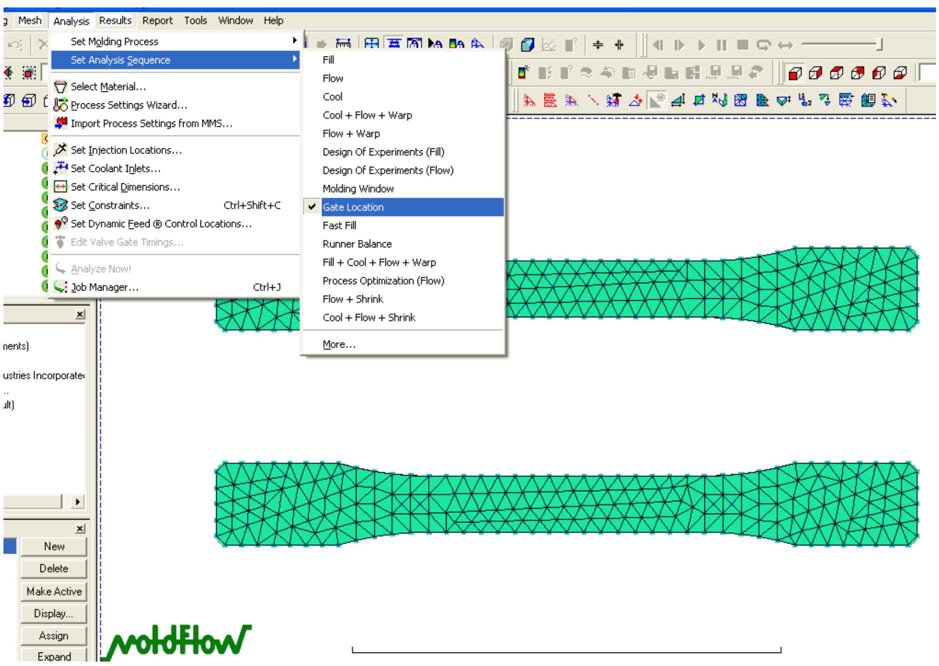


Figure E 10

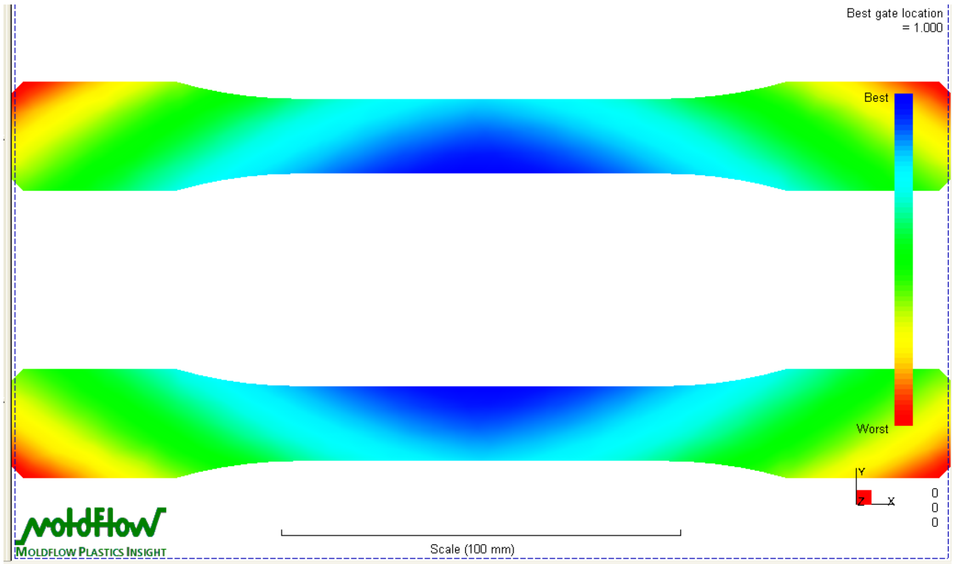


Figure E 11

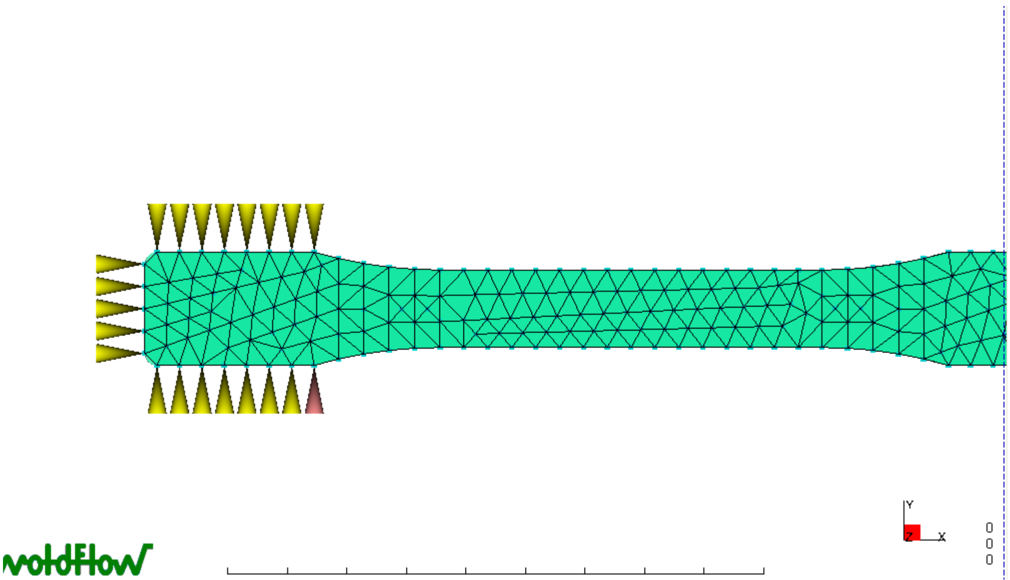


Figure E 12

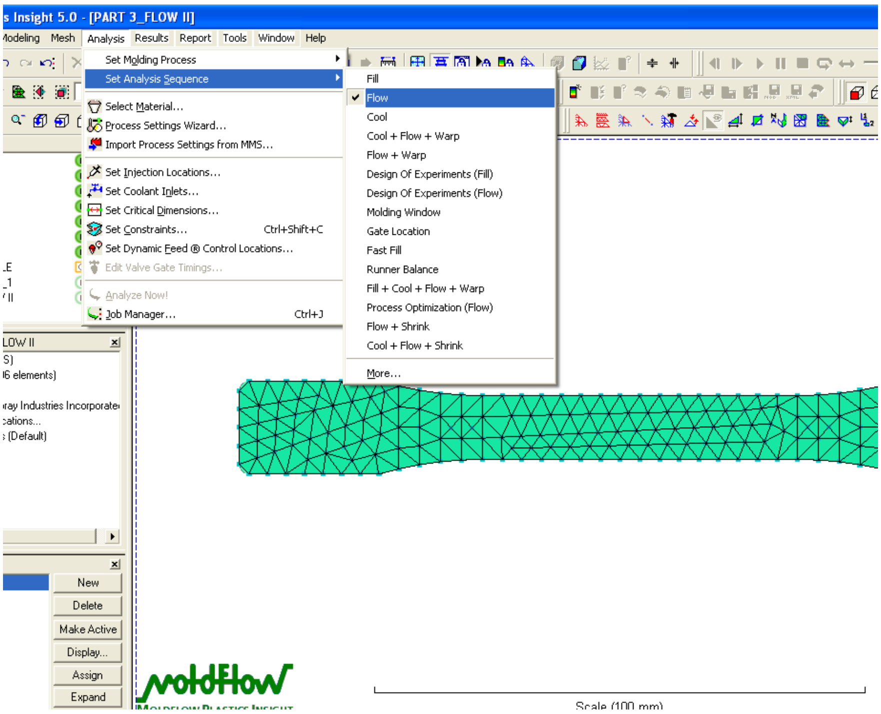


Figure E 13

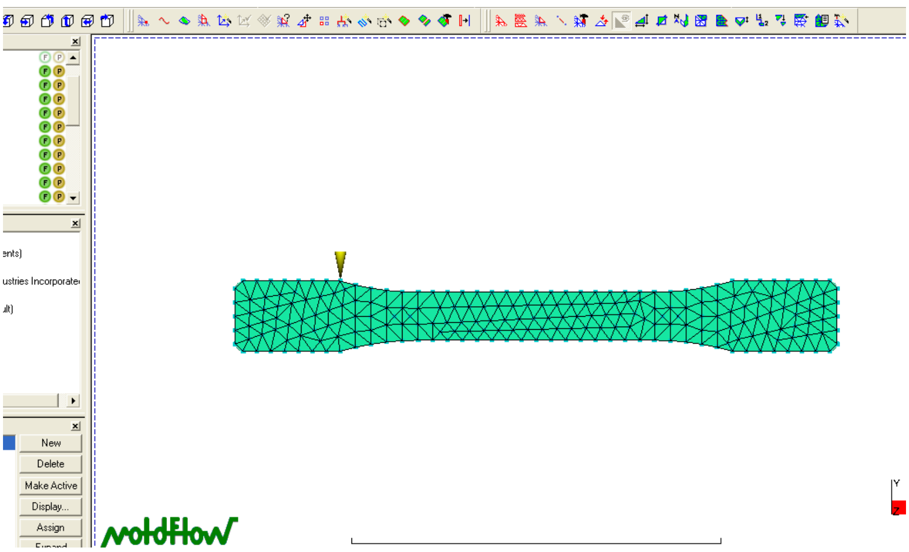


Figure E 14

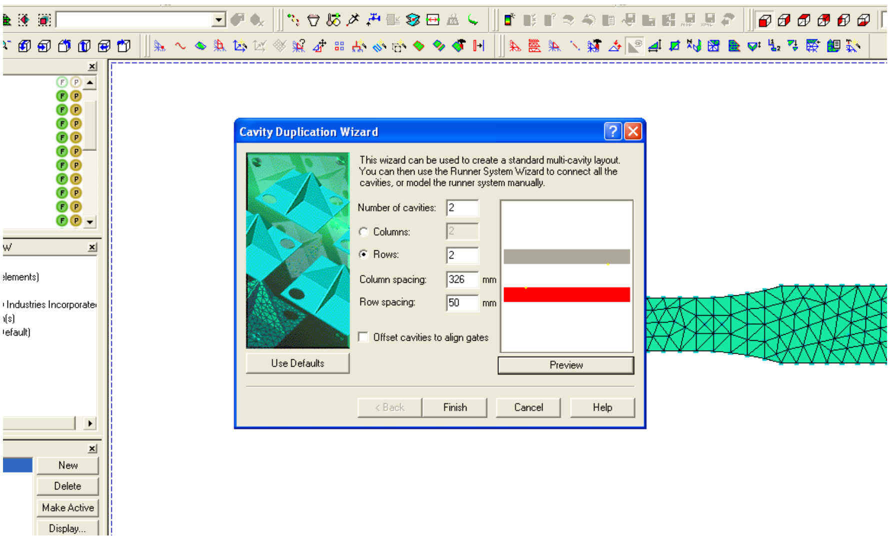


Figure E 15

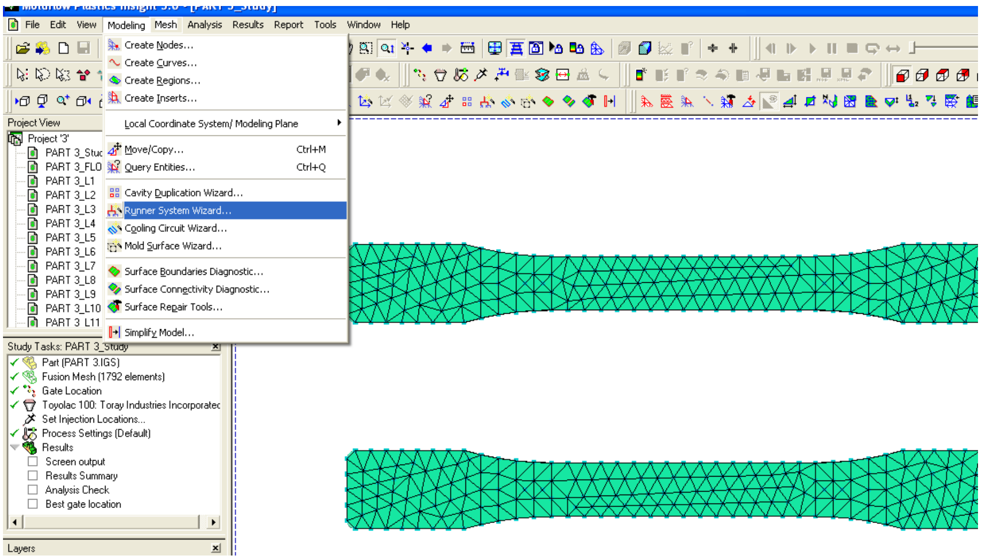


Figure E 16

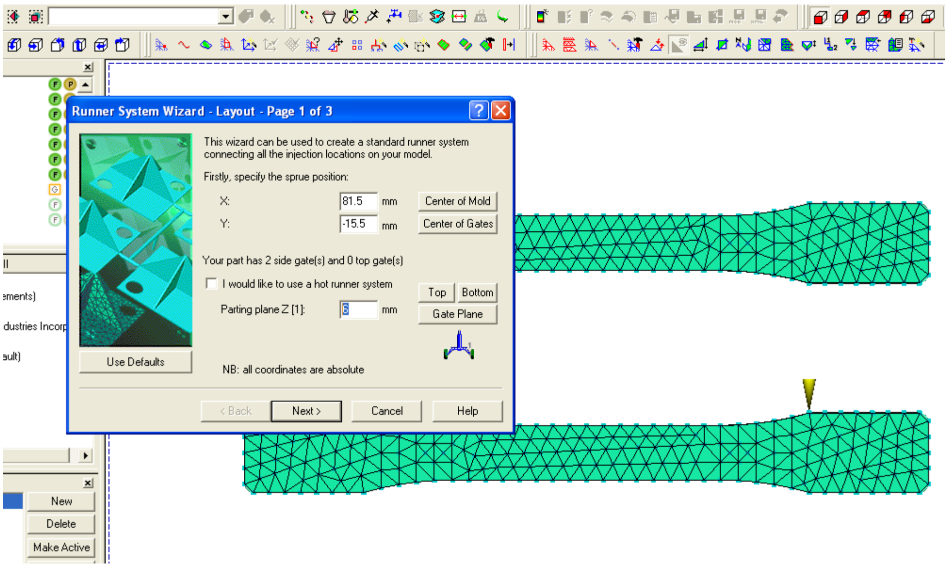


Figure E 17

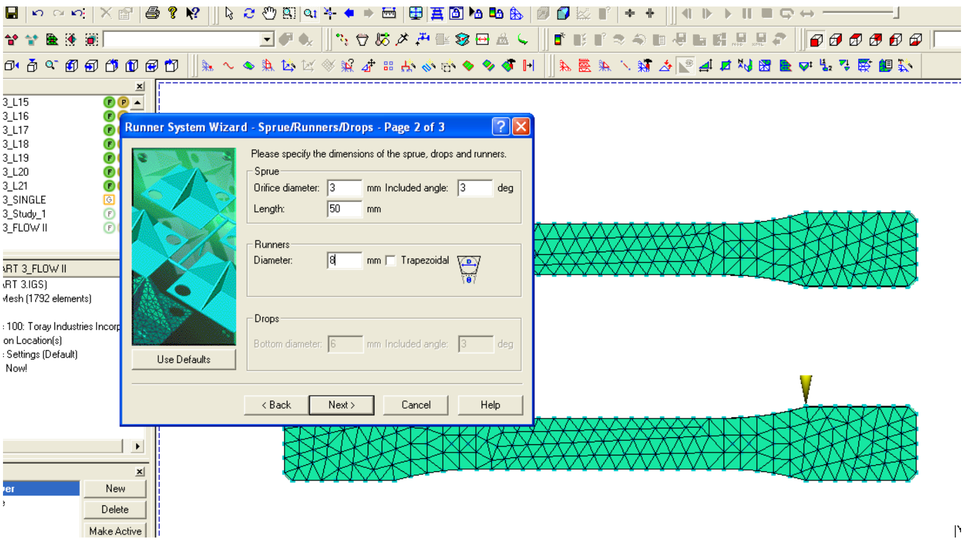


Figure E 18

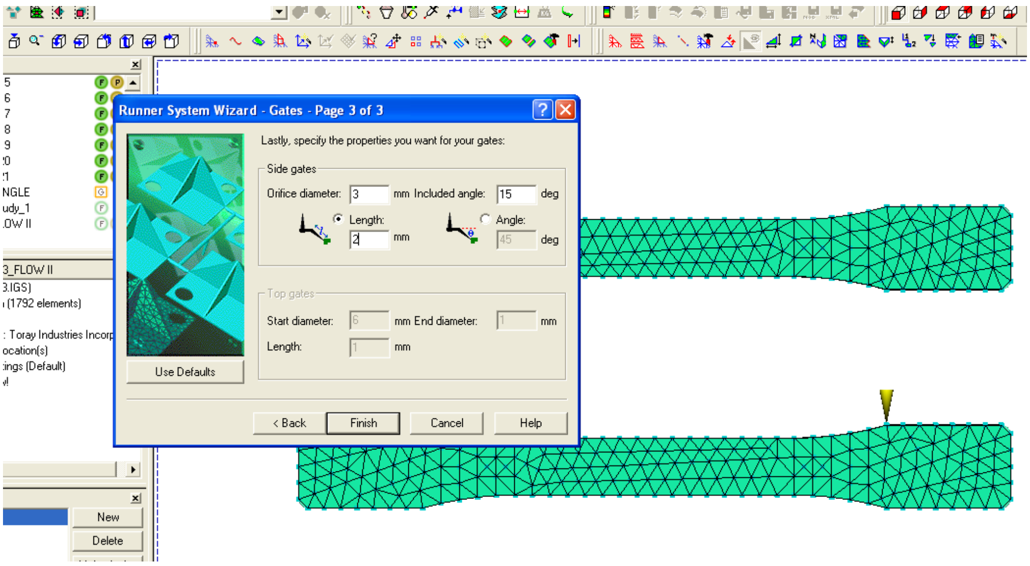


Figure E 19

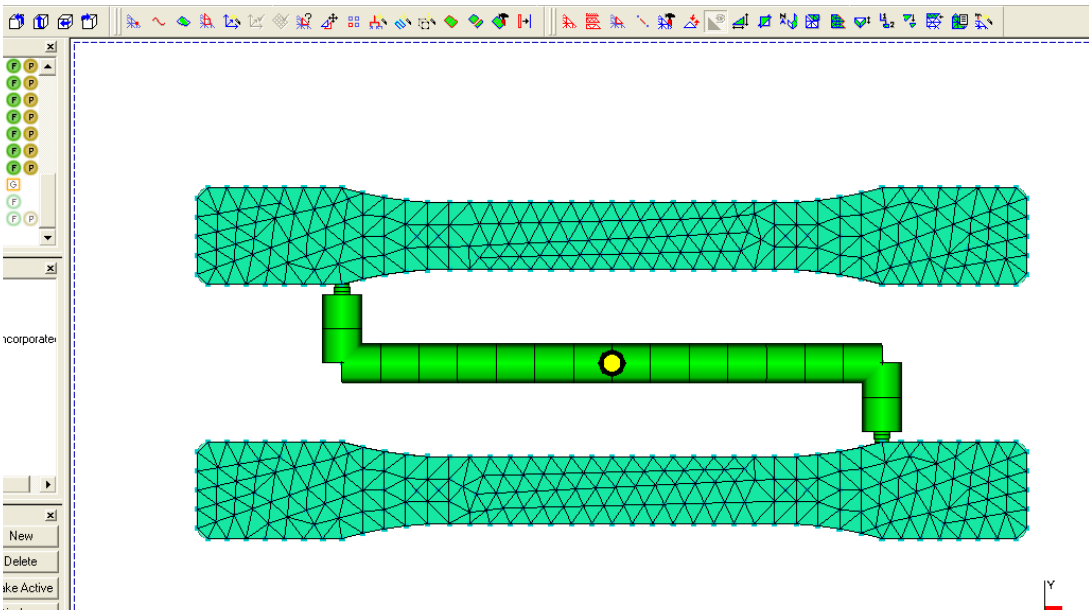


Figure E 20

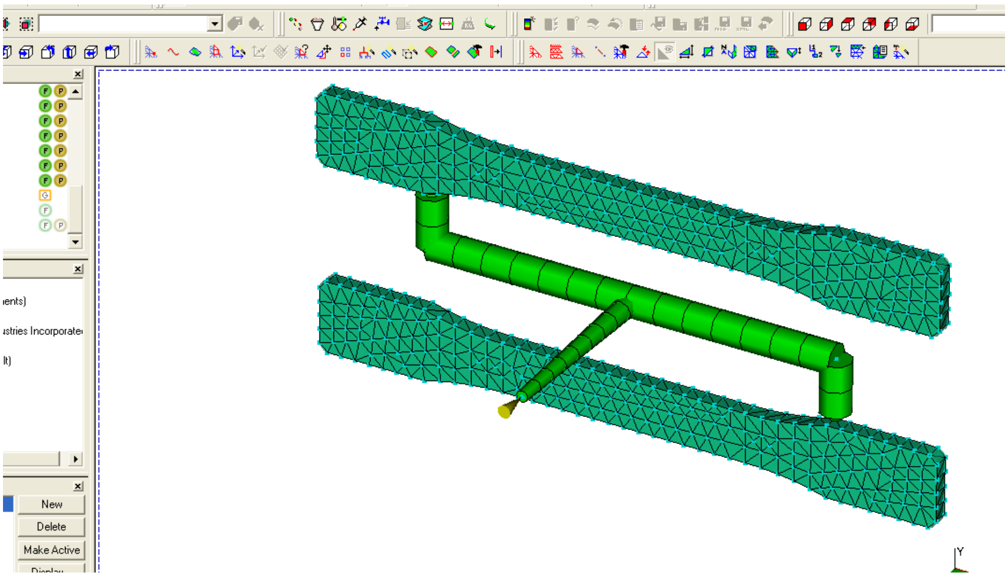


Figure E 21

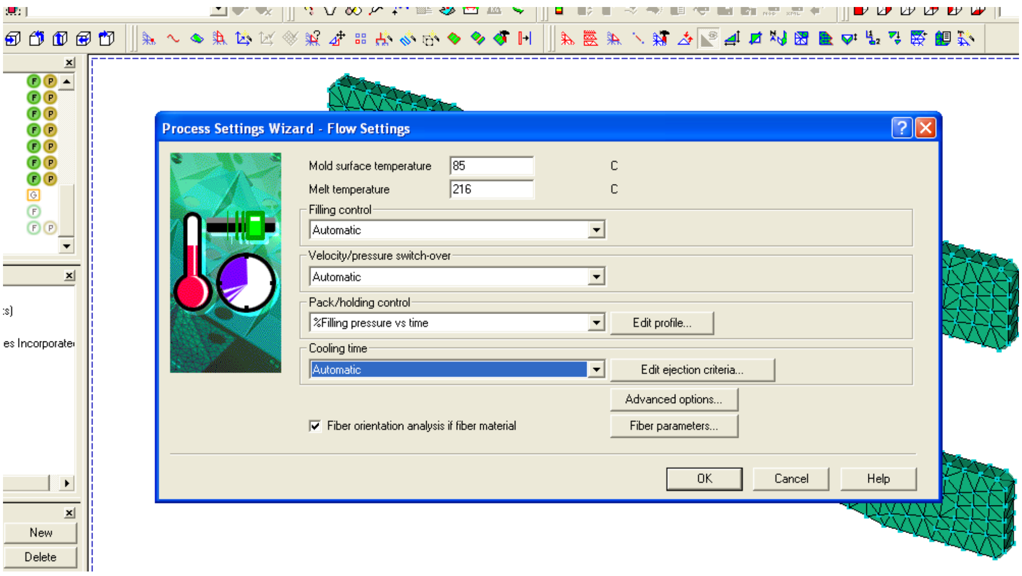


Figure E 22

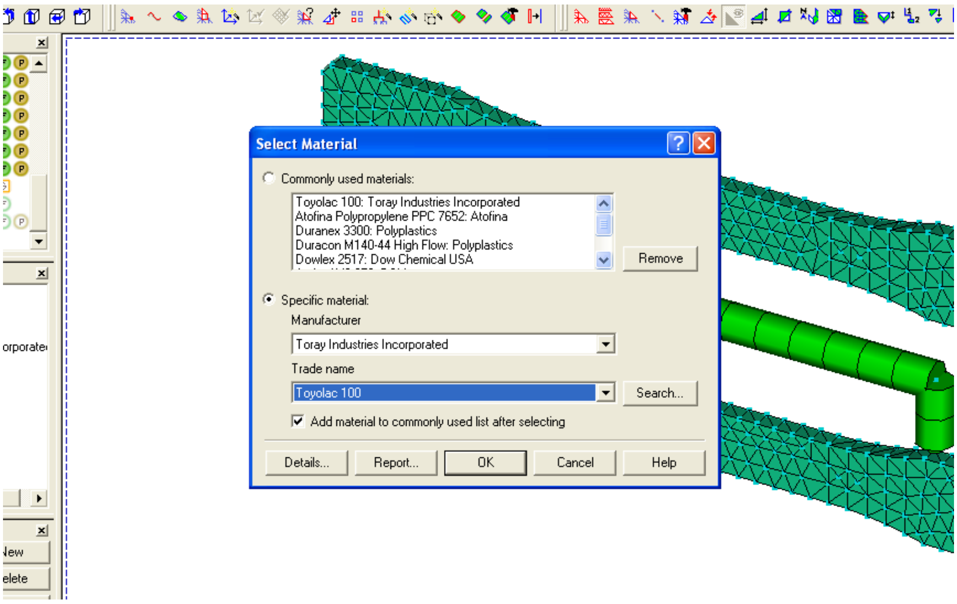


Figure E 23

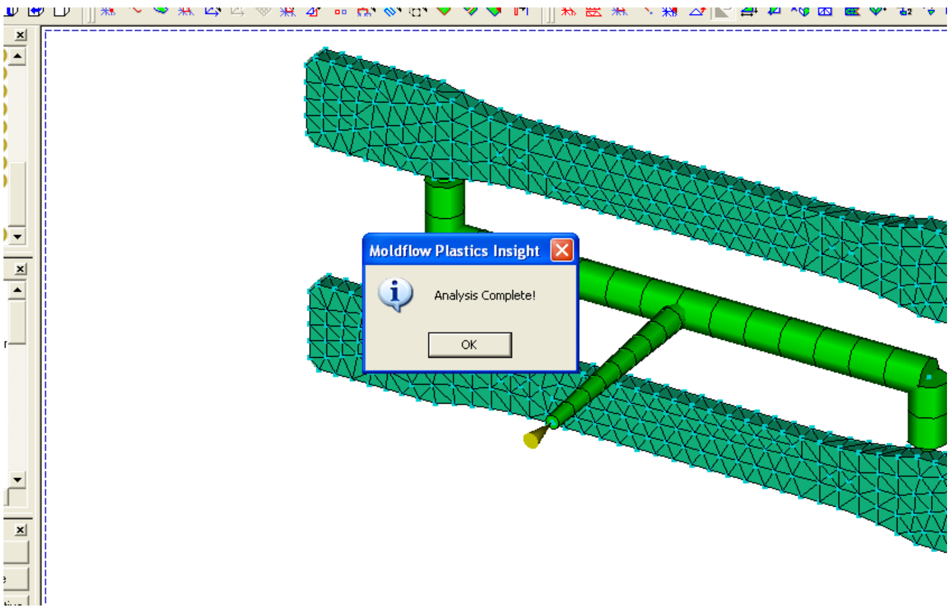


Figure E 24

APPENDIX F

Type of Gate

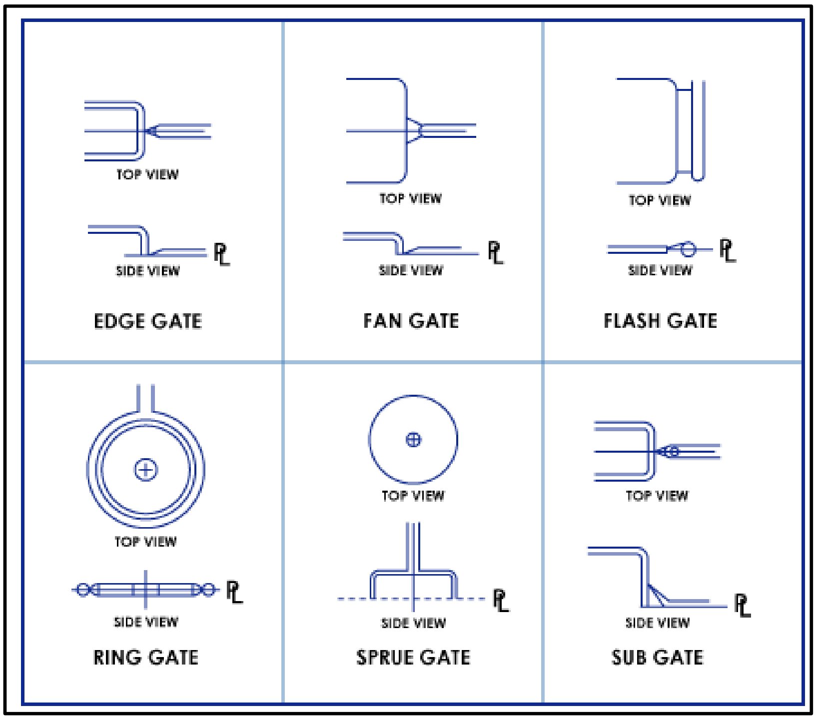


Figure F1: Common gates type

Source: (C.E. Alfredo. 2006)

APPENDIX G

Table G1: Runner cross-section characteristics

Cross-section	Descriptions	
a/ Full Round	Advantages	Smallest surface relative to cross-section, slowest cooling rate, low heat and frictional losses, center of channel freezes last therefore effective holding pressure.
	Disadvantages	Machining into both mold half is difficult and expensive.
b/ Trapezoidal	Advantages	Best approximation of circular cross-section, simpler machining in one mold half only (moveable half)
	Disadvantages	More heat losses and scrap compare to circular cross-section
c/ Half Round	Unfavorable cross-sections and have to be avoided	

Source: (Hanser, 2001)

APPENDIX H

RESULTS ANALYSIS

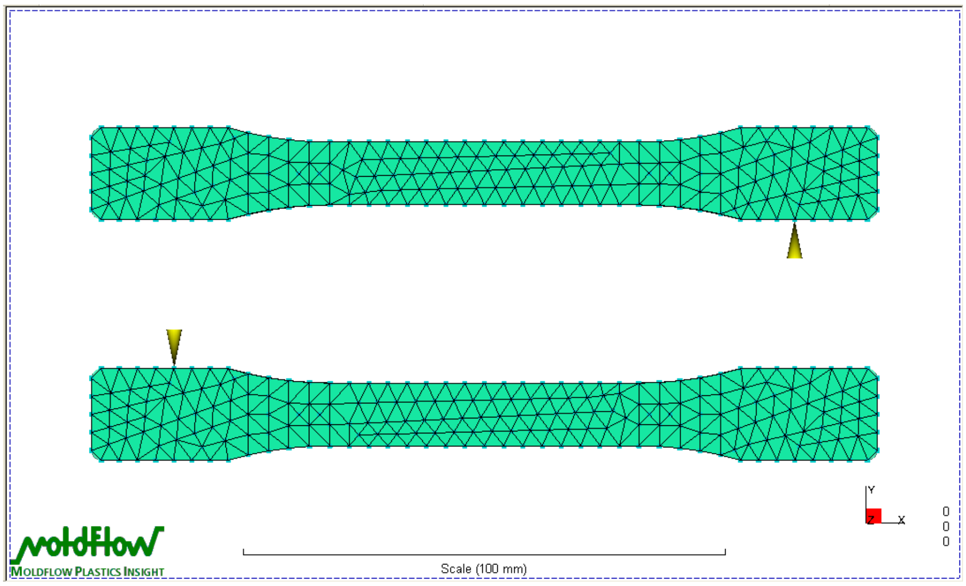


Figure H 1: Location 4(16mm from origin location)

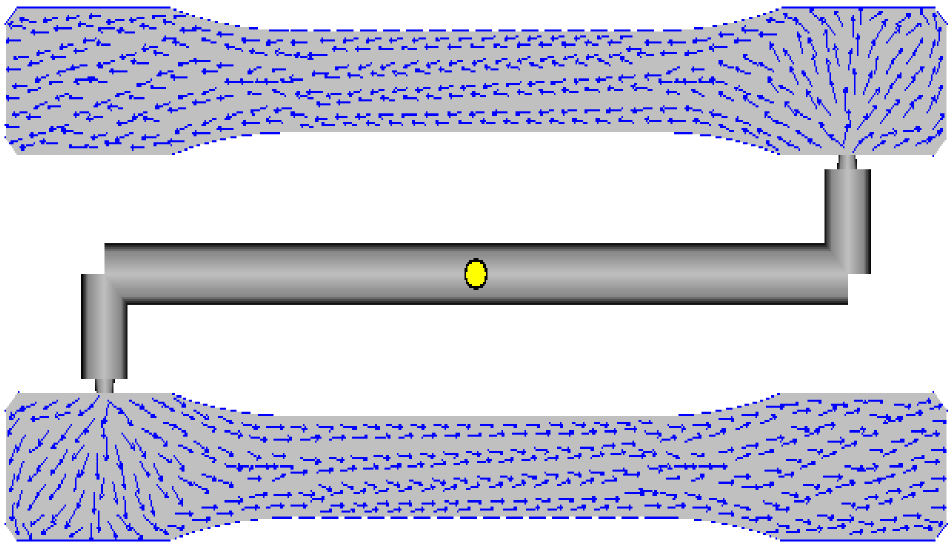


Figure H 2: Orientation at location 4

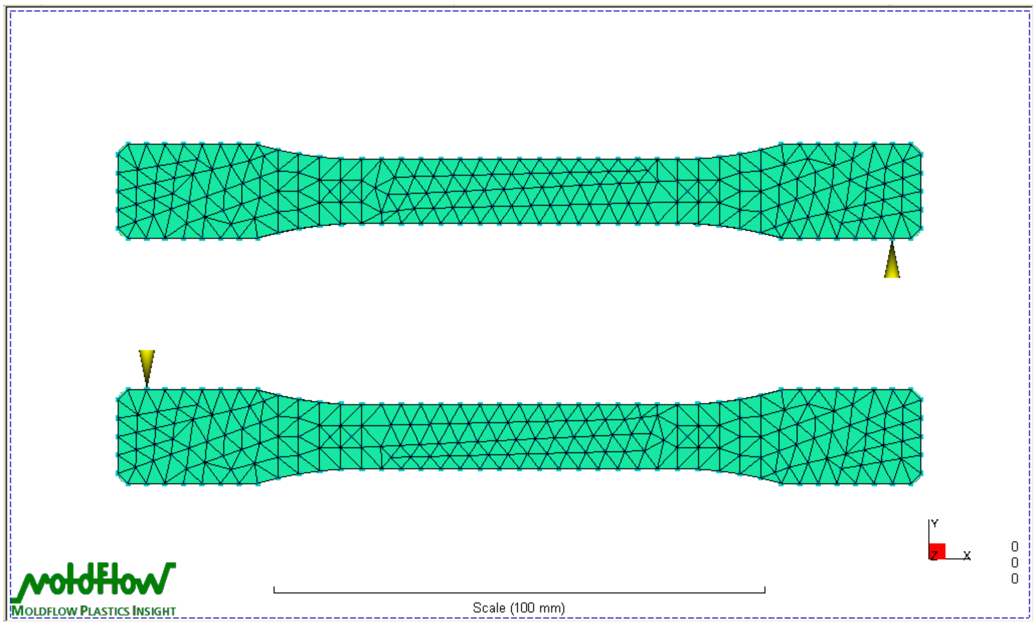


Figure H 3: Location 7(28mm from origin location)

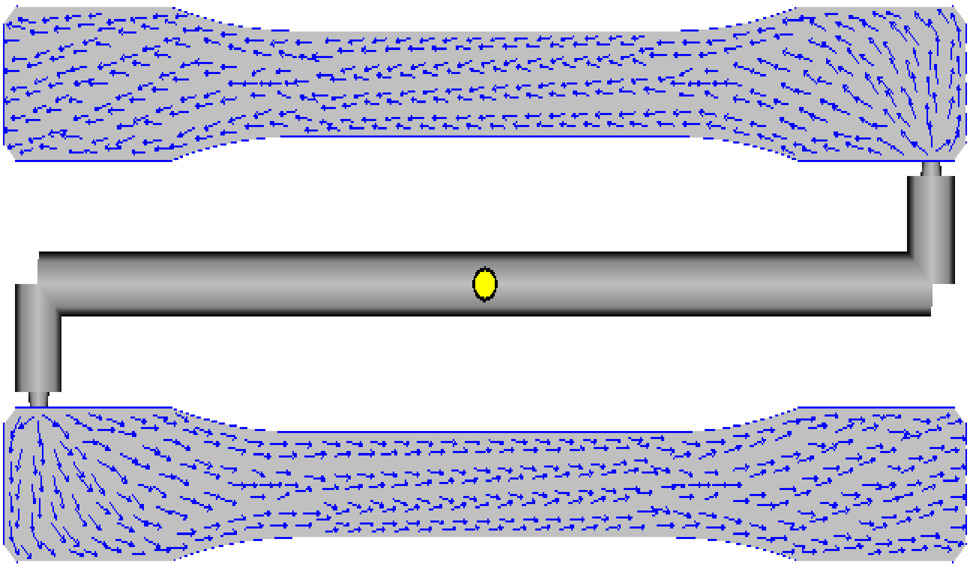


Figure H 4: Orientation at location 7

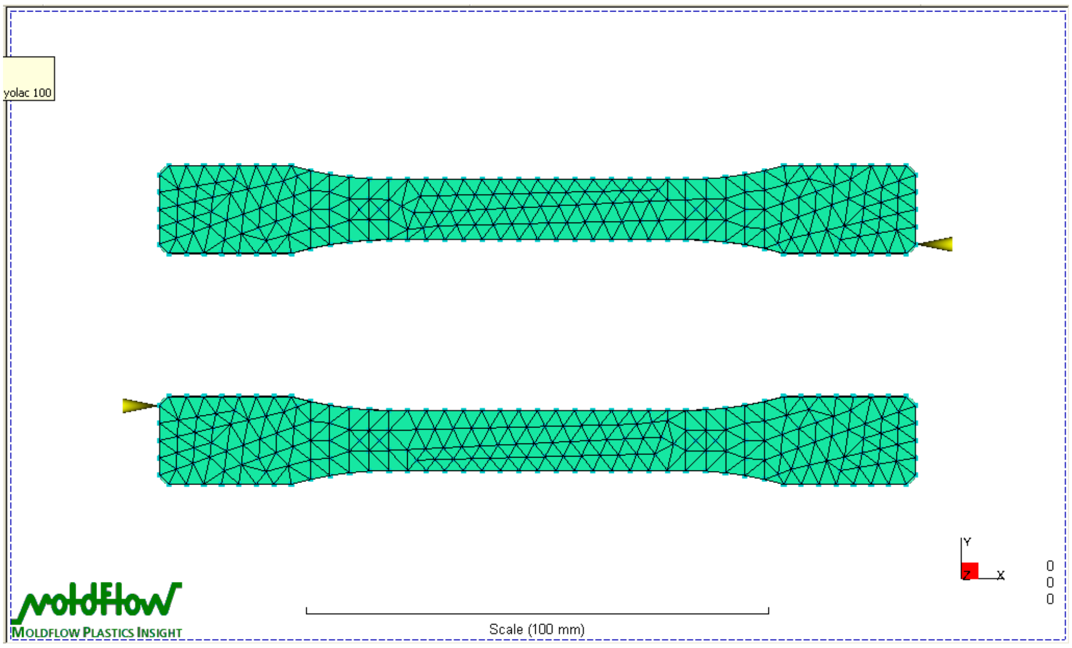


Figure H 5: Location 9(36mm from origin location)

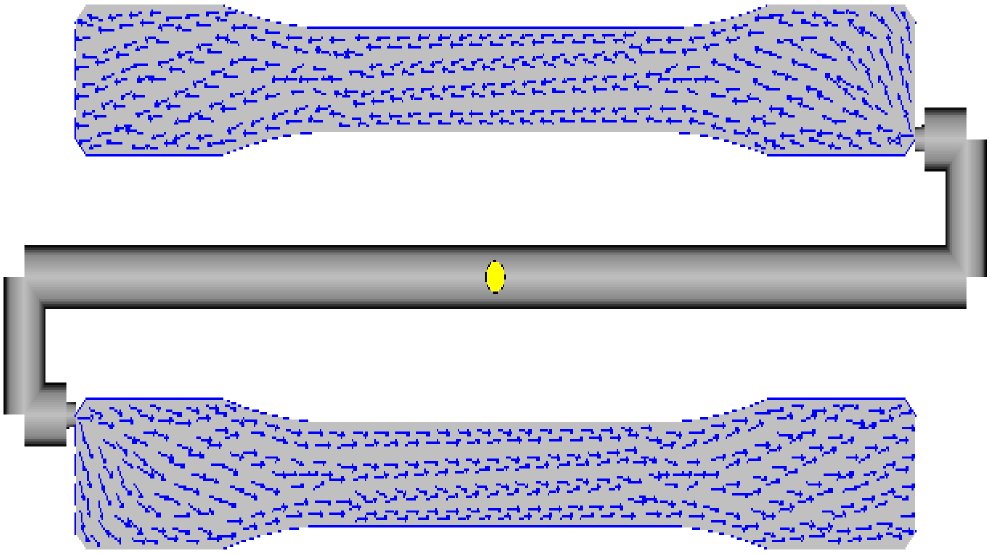


Figure H 6: Orientation at location 9

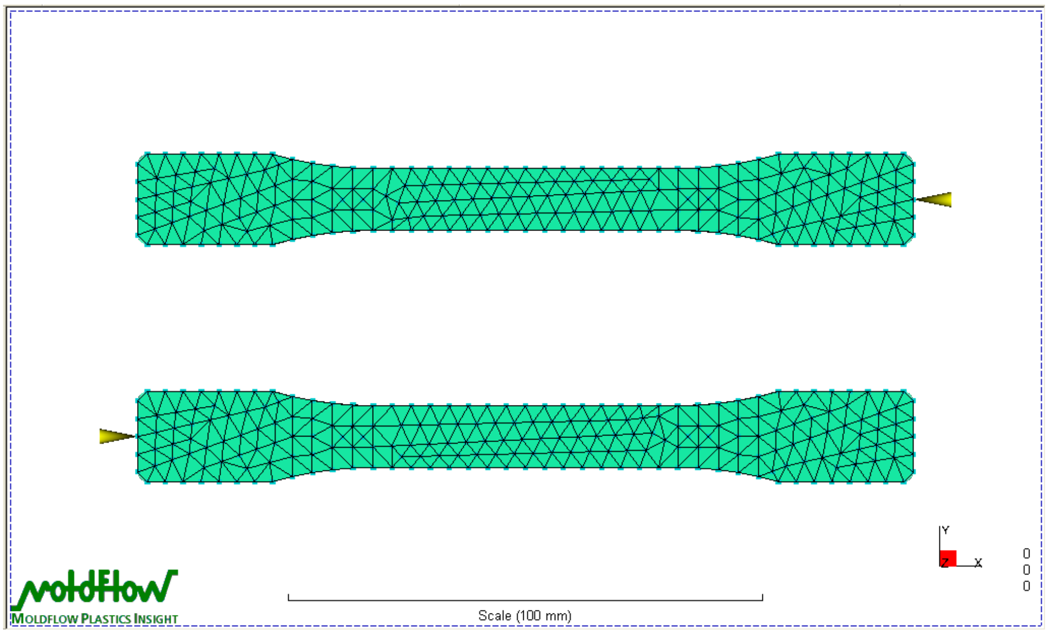


Figure H 7: Location 11(44mm from origin location)

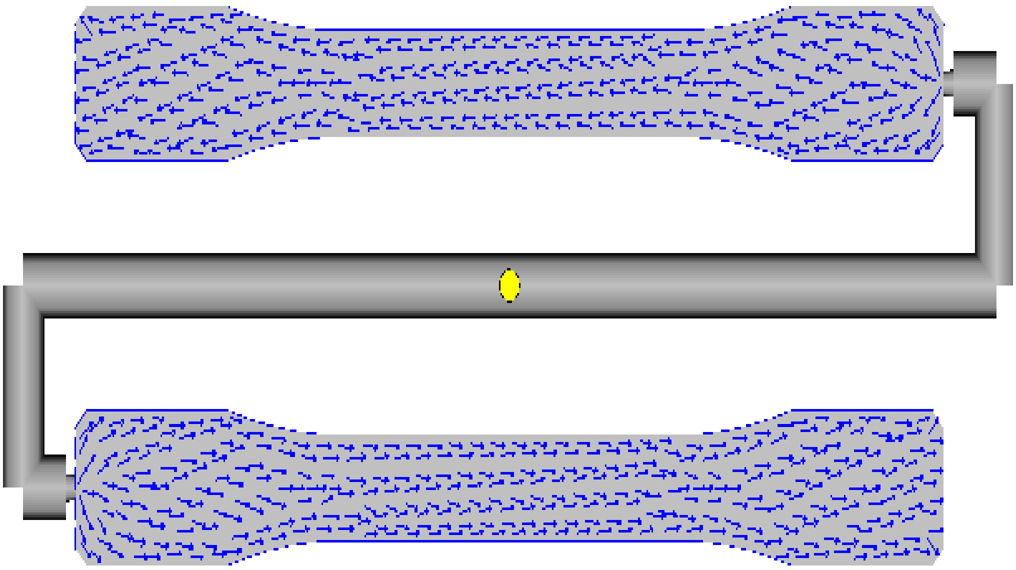


Figure H 8: Orientation at location 11

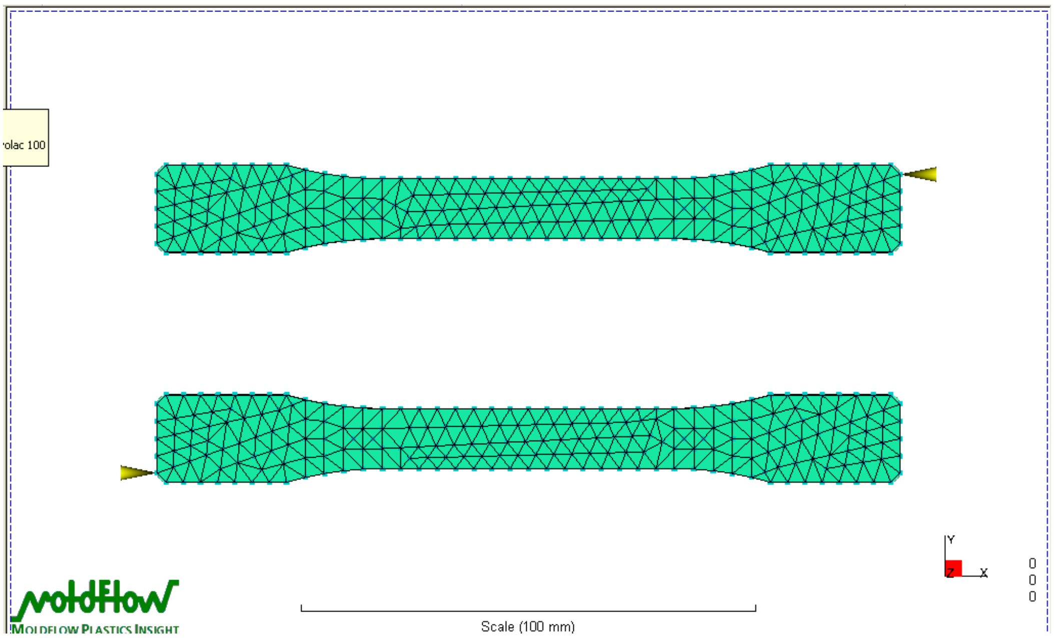


Figure H 9: Location 13(52mm from origin location)

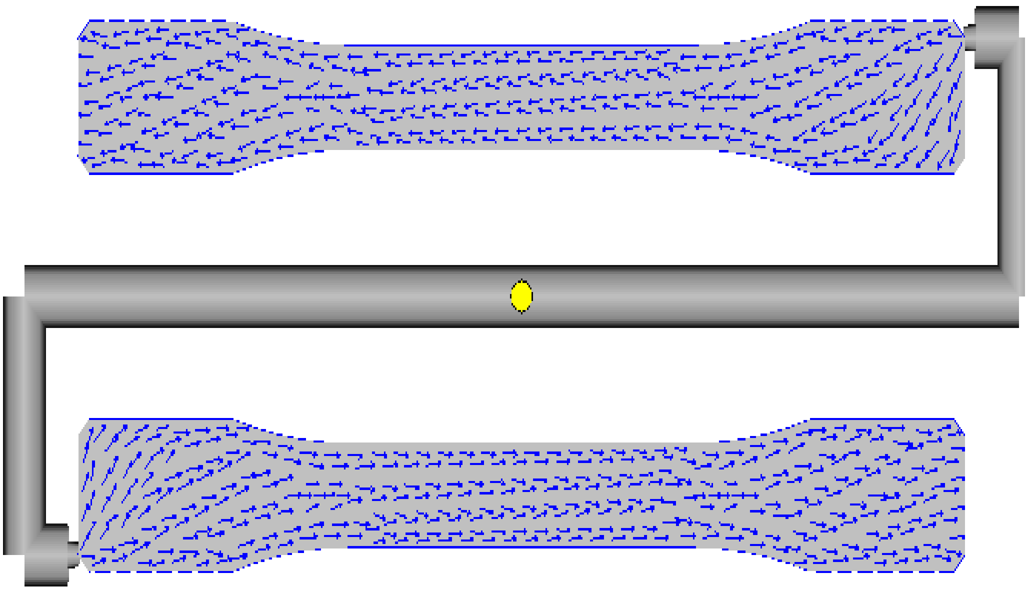


Figure H 10: Orientation at location 13

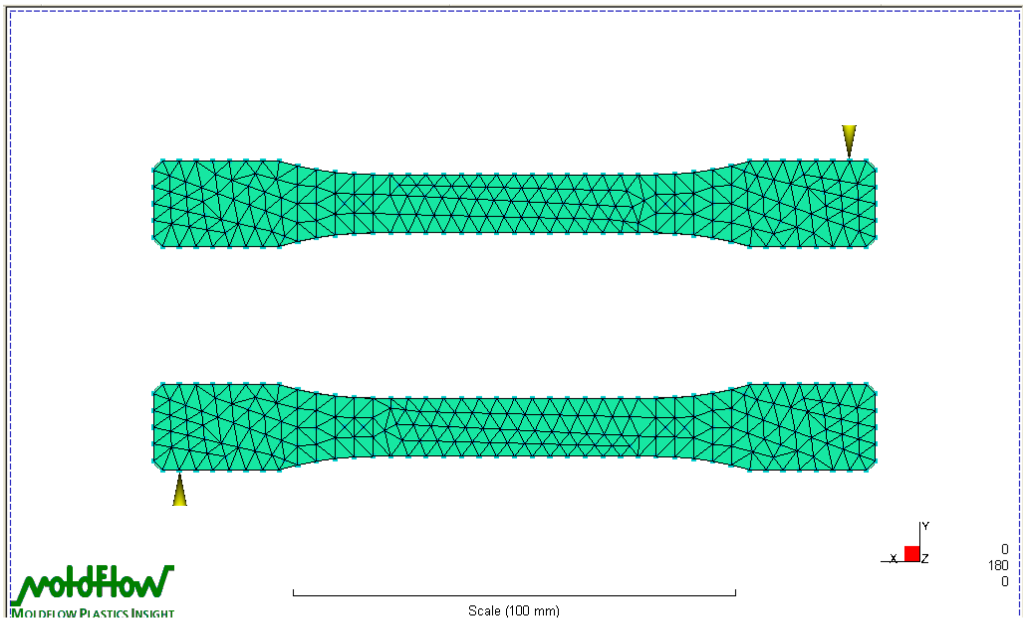


Figure H 11: Location 15(60mm from origin location)

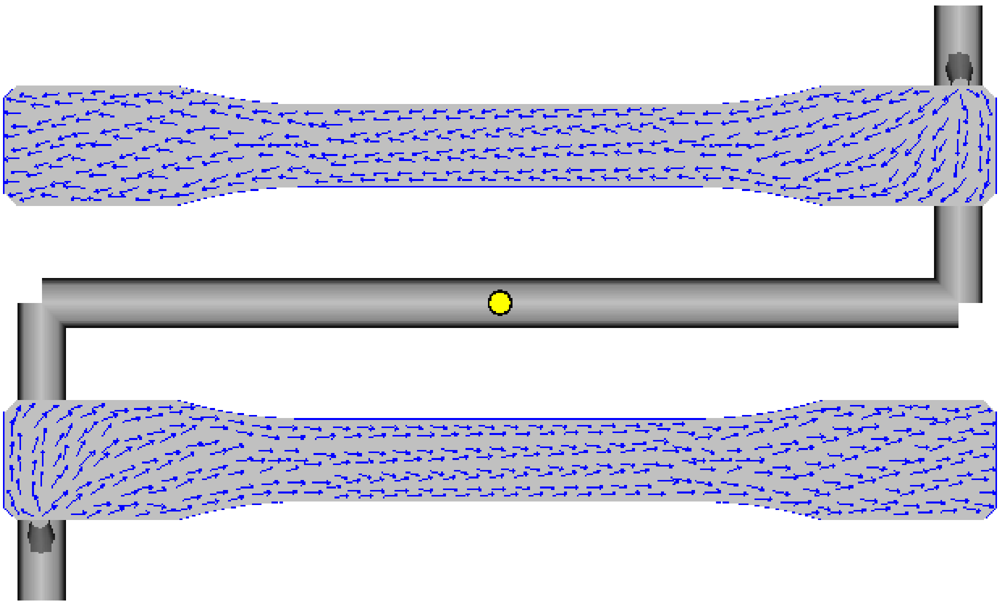


Figure H 12: Orientation at location 15

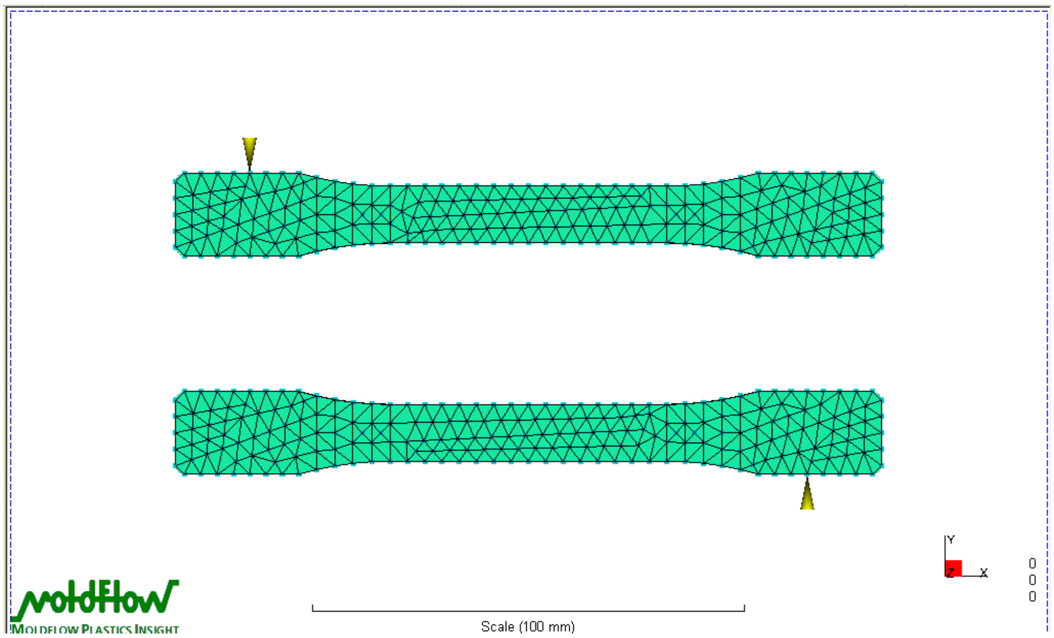


Figure H 13: Location 18(72mm from origin location)

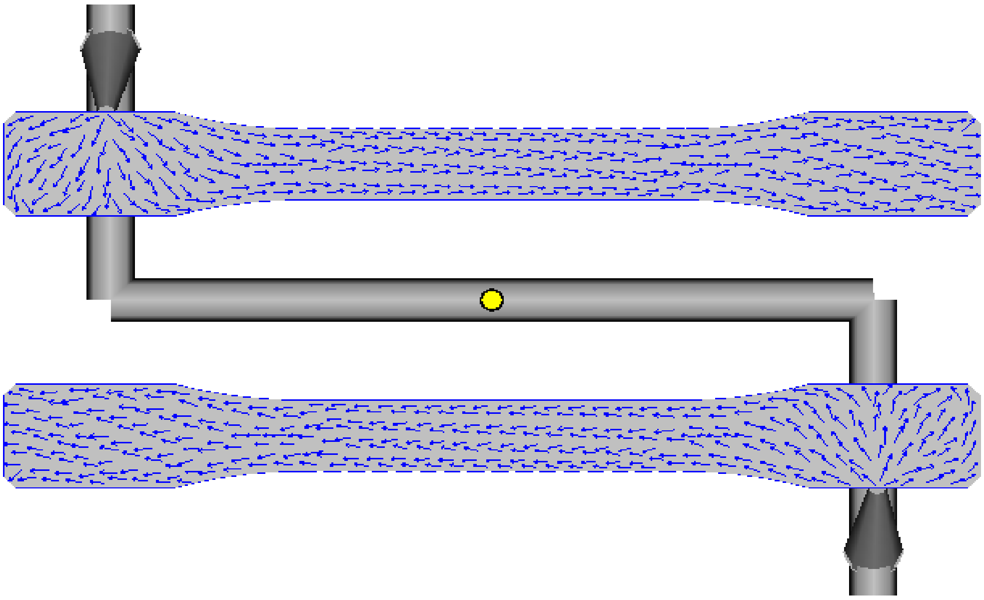


Figure H 14: Orientation at location 18

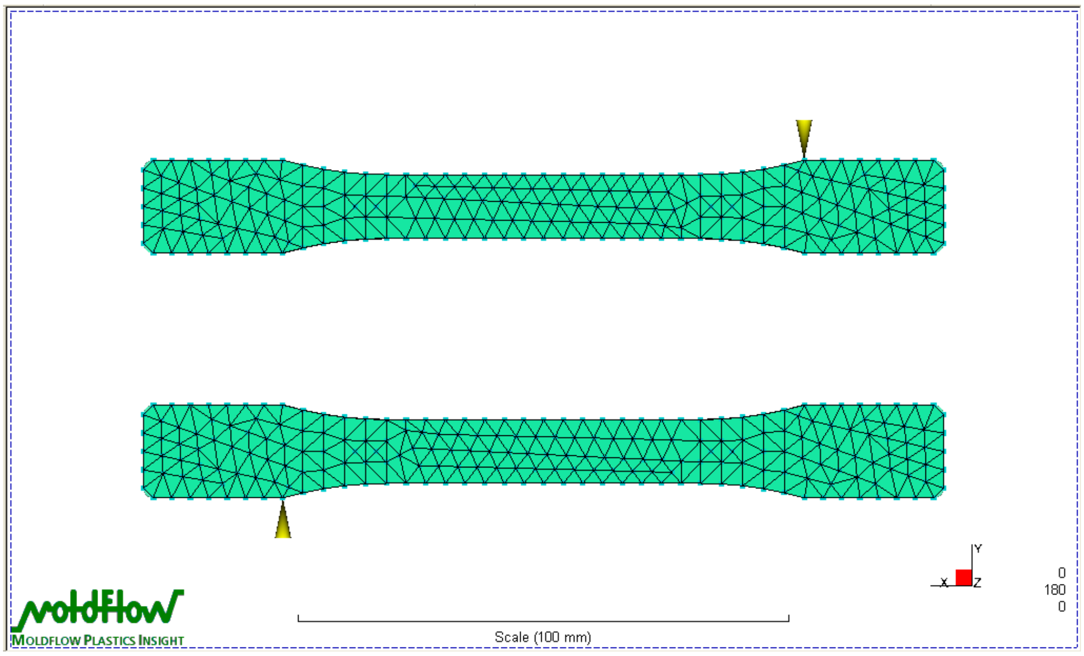


Figure H 15: Location 21(84mm from origin location)

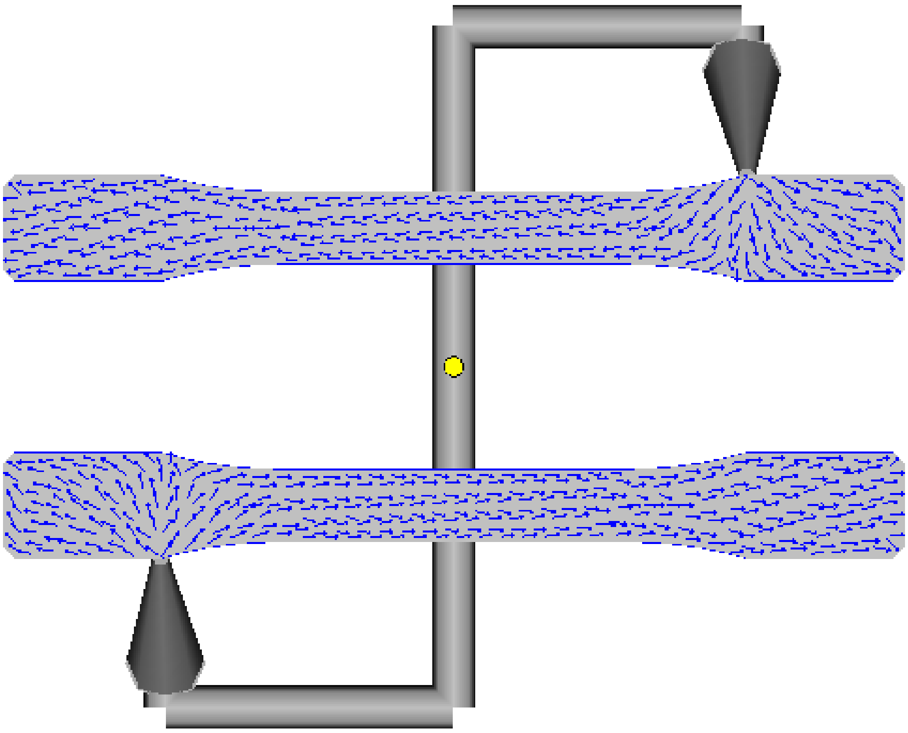
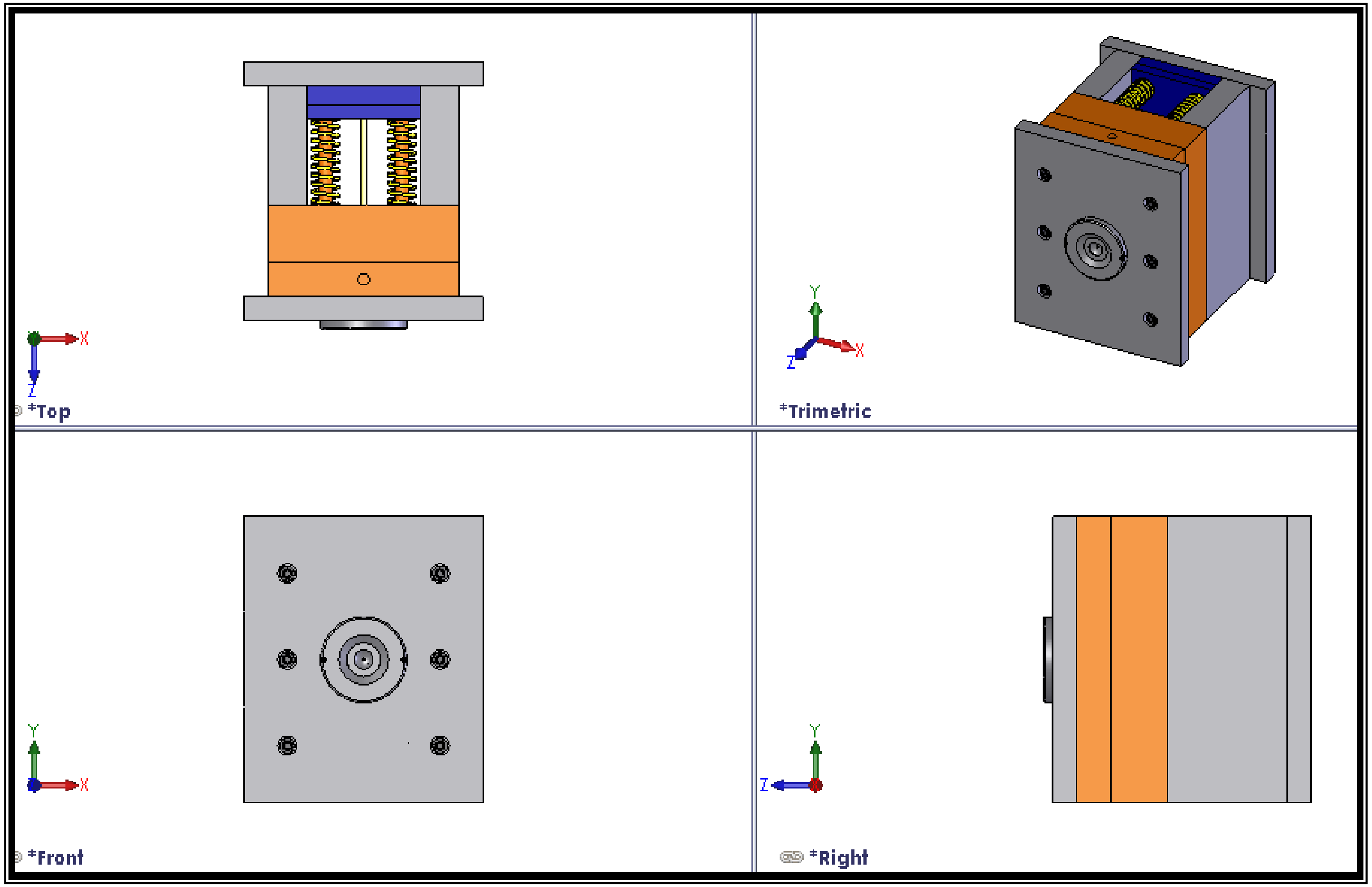


Figure H 16: Orientation at location 21

I 1 : Three view drawing of Two Plate Molds



APPENDIX I

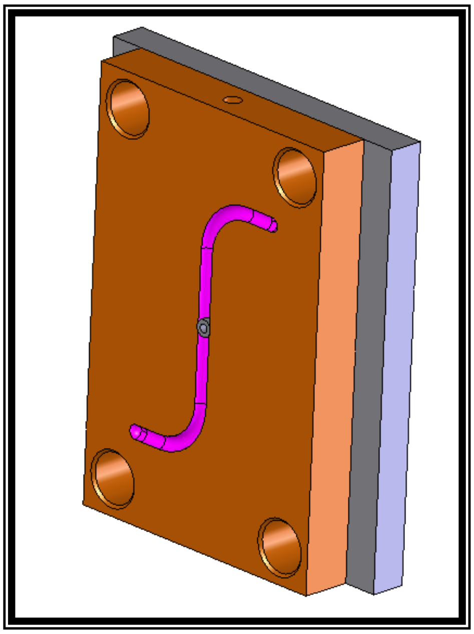


Figure I 2: Half A Isometric view

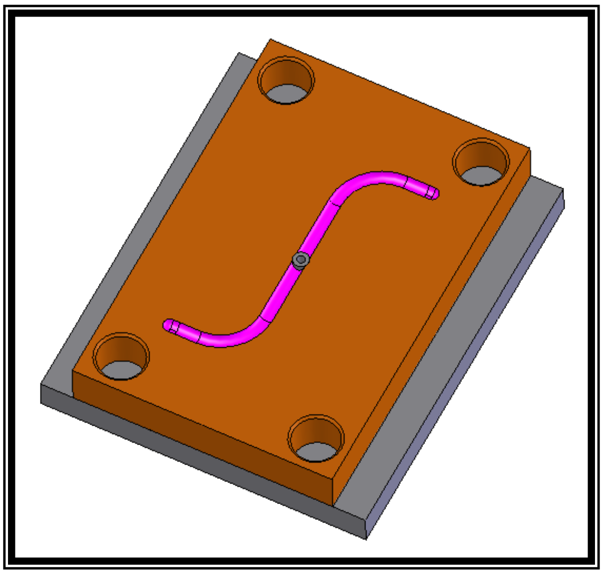


Figure I 3: Half A with runner construction

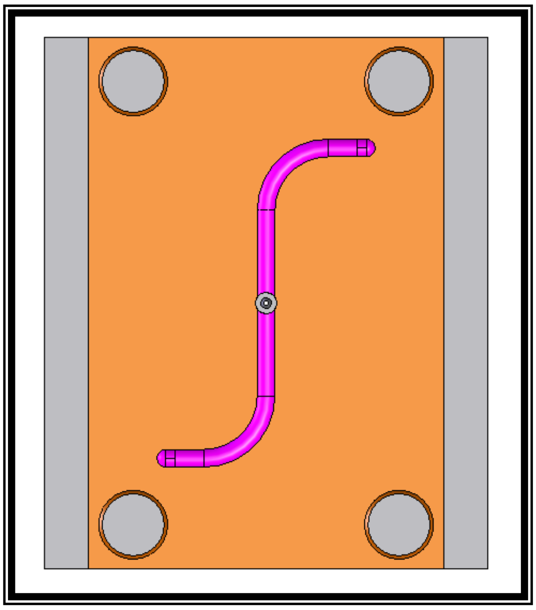


Figure I 4: Front view of Half A

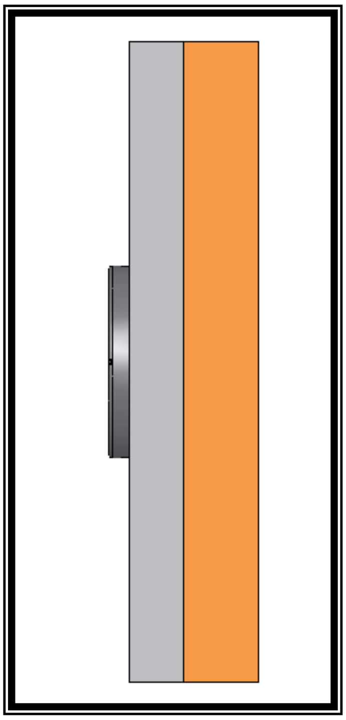


Figure I 5: Side view of Half A

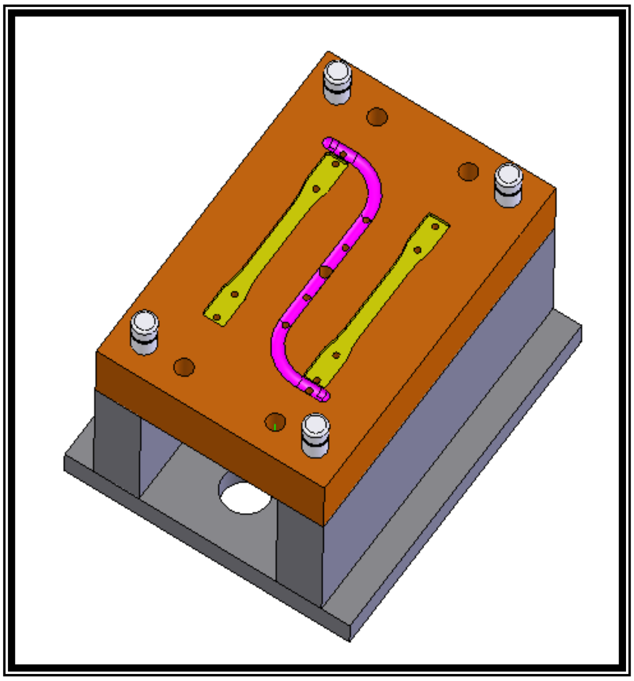


Figure I 6: Half B with runner and insert construction

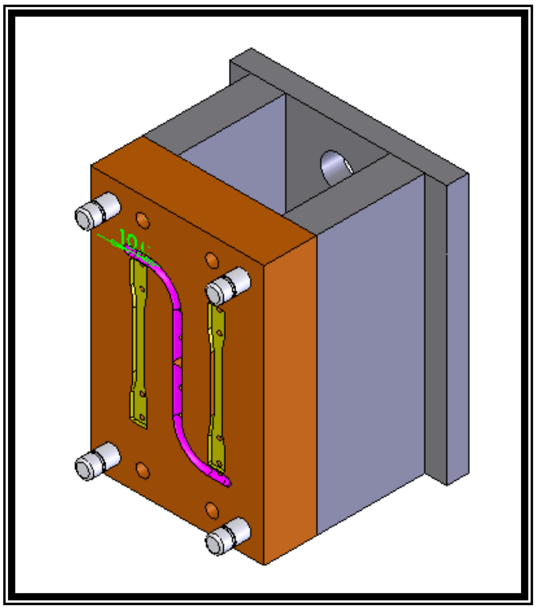


Figure I 7: Half B Isometric view

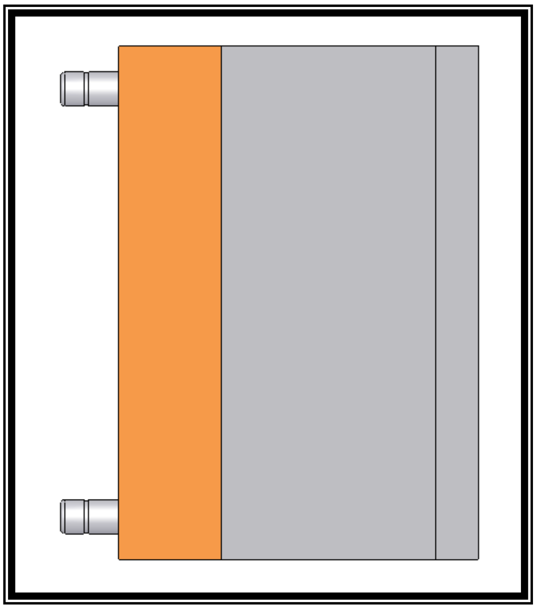


Figure I 8: Half B side view

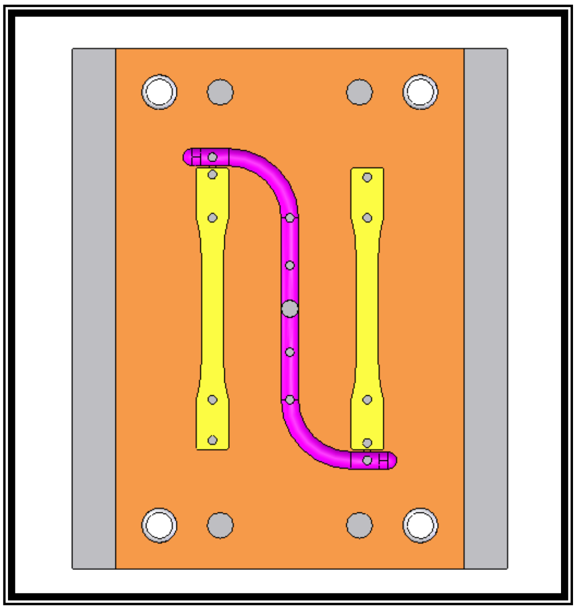


Figure I 9: Half B Front view

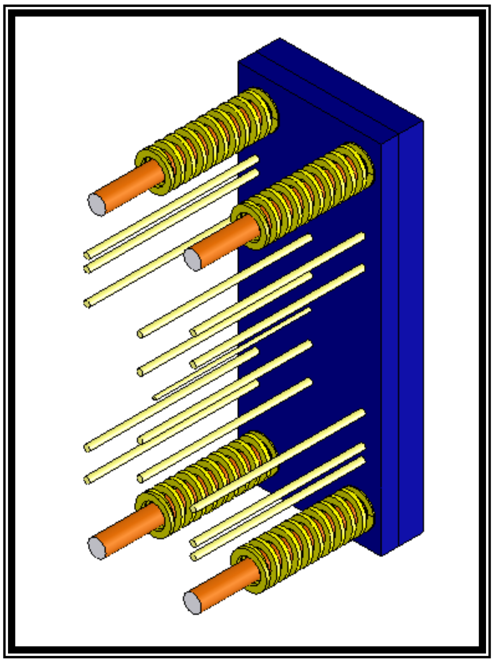


Figure I 10: Ejector system isometric view

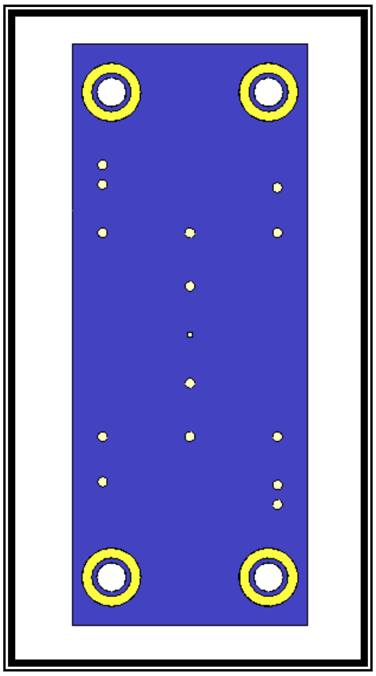


Figure I 11: Front view of ejector system

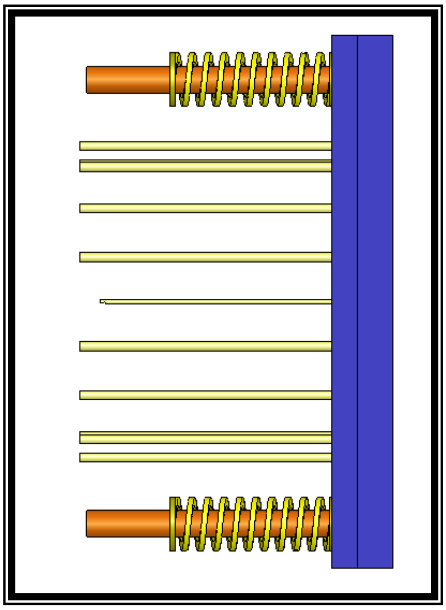


Figure I 12: Side view of ejector system

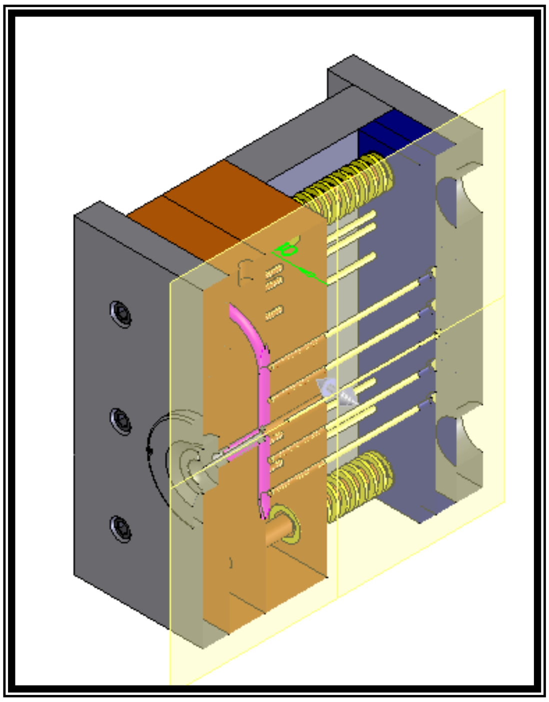


Figure I 13: Isometric cross-section view of complete Two Plate Molds design

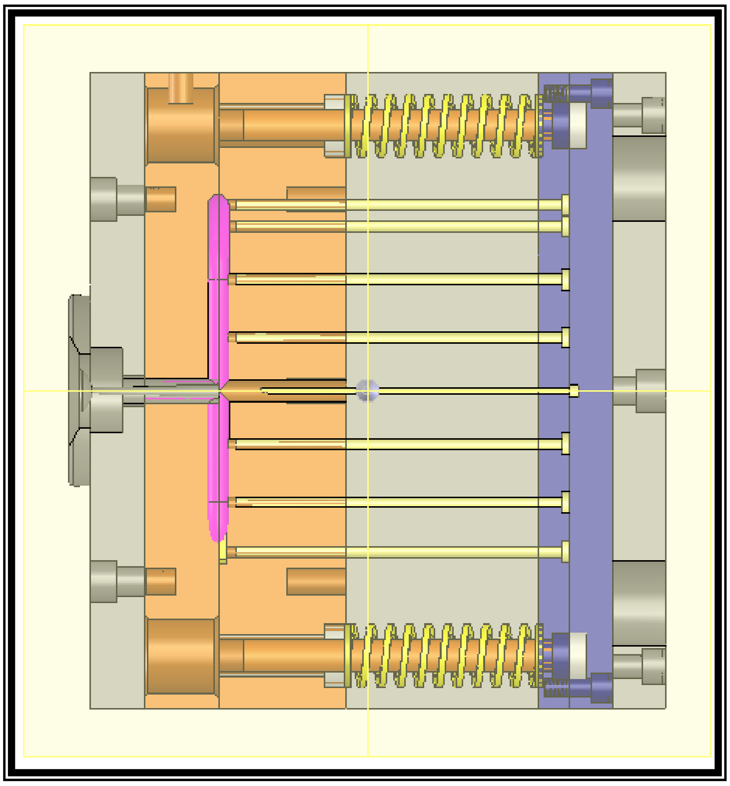


Figure I 14: Cross-Section of side view

APPENDIX J

TENSILE TESTING OF PLASTICS

ASTM D638, ISO 527

A2LA Accredited
(http://www.ptli.com/testlopedia/tests/tensile-plastics-D638-ISO527.asp)

Scope:

Tensile tests measure the force required to break a specimen and the extent to which the specimen stretches or elongates to that breaking point. Tensile tests produce a stress-strain diagram, which is used to determine tensile modulus. The data is often used to specify a material, to design parts to withstand application force and as a quality control check of materials. Since the physical properties of many materials (especially thermoplastics) can vary depending on ambient temperature, it is sometimes appropriate to test materials at temperatures that simulate the intended end use environment.

Test Procedure:

Specimens are placed in the grips of the Instron at a specified grip separation and pulled until failure. For ASTM D638 the test speed is determined by the material specification. For ISO 527 the test speed is typically 5 or 50mm/min for measuring strength and elongation and 1mm/min for measuring modulus. An extensometer is used to determine elongation and tensile modulus.

Elevated or Reduced Temperature Test Procedure:

A thermal chamber is installed on the Instron universal test machine. The chamber is designed to allow the test mounts from the base and crosshead of the Instron to pass through the top and bottom of the chamber. Standard test fixtures are installed inside the chamber, and testing is conducted inside the controlled thermal environment the same as it would be at ambient temperature. The chamber has internal electric heaters for elevated temperatures and uses external carbon dioxide gas as a coolant for reduced temperatures. The size of the chamber places a limitation on the maximum elongation that can be reached, and extensometers are generally limited to no more than 200° C.

Specimen size:

The most common specimen for ASTM D638 is a Type I tensile bar. The most common specimen for ISO 527 is the ISO 3167 Type 1A multipurpose specimen. ASTM D882 uses strips cut from thin sheet or film.

Data:

The following calculations can be made from tensile test results:

1. tensile strength (at yield and at break)
2. tensile modulus
3. strain
4. elongation and percent elongation at yield
5. elongation and percent elongation at break

Equipment Used:

Instron Universal Tester Extensometers