

APSX-PIM BEGINNER'S GUIDE TO INJECTION MOLD DESIGN



COMPACT Guidelines to
Make Injection Molds for
the APSX-PIM Injection
Machine



Foreword

At APSX, our mission is to help customers unlock the full versatility of injection molding—from the earliest concept sketches to the very last service-life replacement part. While our machines are engineered for rapid, reliable production, every successful shot ultimately depends on the quality of the mold. The right mold-design strategy can spell the difference between a smooth product launch that scales effortlessly and a costly, reputation-damaging recall.

This guide is not intended to be an exhaustive treatise on mold design. Instead, it distills proven best practices we have seen propel customers to success with APSX-PIM and highlights key insights we have gathered alongside partners throughout an ever-evolving industry.

- CAD Chapters: Developed with the expertise and real-world perspectives of APSX LLC.
- 3D-Printed Mold Coverage: Informed by the teams at Formlabs and 3DPMolds.

If you spot a gap or have a case study that could inspire others, we would love to hear from you. Drop us a note at info@apsx.com, and let's advance the craft of injection molding together.

1 - Introduction

The aluminum molds are an indispensable part of product development journey. The design and procurement of traditional steel molds may tie development teams into 6–8-week lead times and five figure machining costs. That is one of the key things we aim to change at APSX. With the APSX-PIM and aluminum molds, you can mill a mold in the morning, shoot parts in the afternoon, revise overnight, and repeat until perfect—without breaking the budget or having to involve external suppliers. This guide walks you through the fundamentals so you can reach that first “good” part faster.

In traditional development and manufacturing workflows, steel mold design and subsequent production may easily consume 6-8 weeks of calendar time and require thousands of dollars in CNC machining costs before the first part is ever produced. This significant delay often forces engineering teams to compromise with a "good-enough" initial design, limiting the number of critical design iterations simply to maintain schedule and comply with budget constraints. The unfortunate result is frequently a compromise: production parts that require costly re-cuts or design concessions that persist in the product for years.

Aluminum Tooling: The APSX-PIM molding system fundamentally transforms this equation. Optimized for compact aluminum tooling, this technology takes a single or multi-cavity molds made from 6061 or 7075 aluminum in a matter of hours using a standard CNC mill (in-house or outsourced) at a fraction of traditional steel tooling costs. Once completed, you can install the mold into the APSX-PIM injection molding press and begin producing parts with production-grade thermoplastics that same day. If dimensional adjustments or gate modifications are needed, you can quickly re-machine the insert, reinstall it, and produce the next revision before the workday ends.

3D Printed Molds: If you need to manage complex part designs, which may require the use of sliders and other mold actions, 3D printed molds may provide a faster and more flexible route to first parts. Molds fitting the APSX-PIM can be printed in a matter of hours allow development teams to become fully self sufficient. When 3D printed prototype molds are combined with rapid aluminum tooling, a seamless design-to-production pipeline is established: utilize 3D printing for early fit-and-function validation of parts and molds, machine aluminum tooling for functional testing and low-volume production runs, and scale to hardened steel only when the design is finalized. When used this way, the APSX-PIM eliminates the "too expensive to iterate" paradigm, enabling engineers to validate geometry, material shrinkage characteristics.

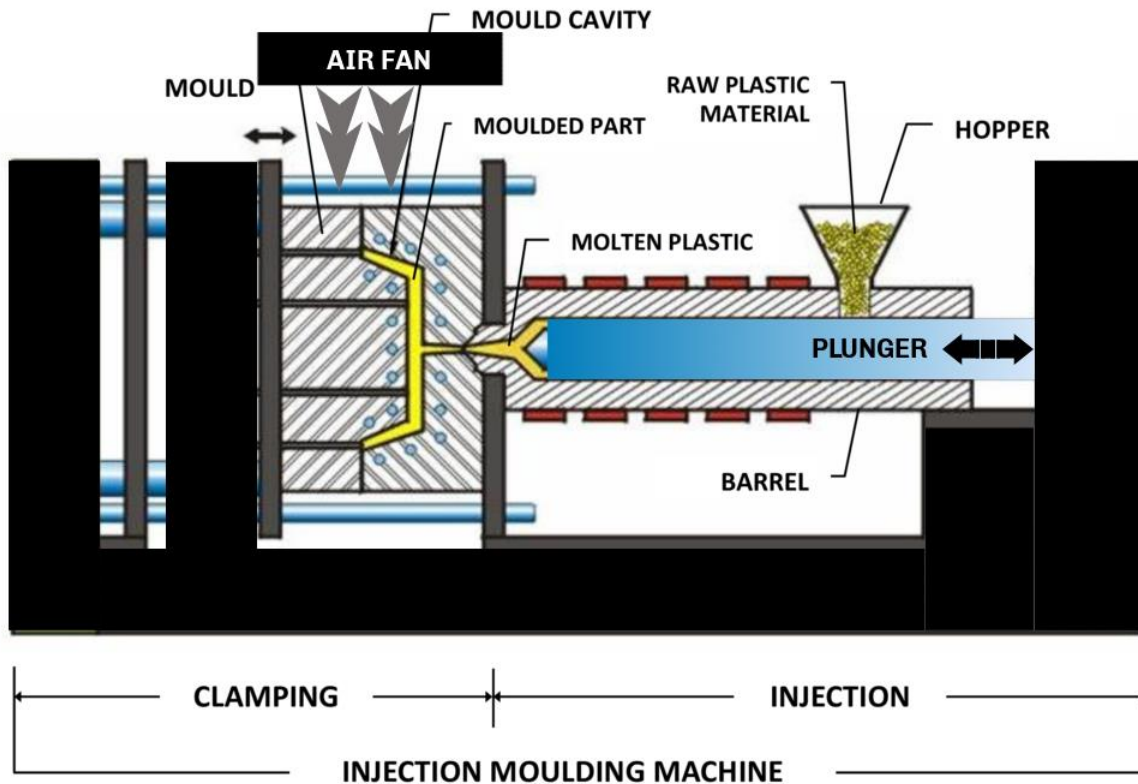
2 - Injection Molding Basics

Injection molding is one of the most prevalent and efficient manufacturing processes for producing polymer components. Consider everyday items such as water bottle caps, smartwatch straps, and sunglass frames. If these products are manufactured from plastic materials, they were almost certainly produced through injection molding.

This document provides a beginner-friendly overview of key design considerations, ensuring strong, functional, and manufacturable injection molded parts. Understanding part and mold design fundamentals is crucial for minimizing defects, reducing costs, and improving part performance.

Here are the basic steps in injection molding:

1. Melt plastic pellets in a heated barrel.
2. Inject the molten polymer into a mold.
3. Pack & hold to compensate for shrinkage.
4. Cool until the part solidifies.
5. Eject or manually pick up the finished part.



APSX-PIM has cartridge heaters embedded inside the barrel. The temperature of the barrel is controlled via the user settings.

The mold temperature is controlled with an air fan placed on top of the mold area.

The holding pressure is set by the user and can be set very low where other industrial machines may not achieve. This feature eliminates the high clamp force requirements to make large parts.

The APSX-PIM has an ejector bearing that allows to use molds with ejector system in MULTI CYCLE MODE. This mode turns the injection molding process into a hands-off operation for high volume production.

3 – Choosing the Right Plastic

RULE 1: You do NOT have to use the ABS. This material has been used by industrial machines because of their fast cooling feature. This allows the plastic part manufacturers to run their extremely expensive machines faster and utilize at their maximum output to be able to pay off those machines quicker.

RULE 2: Select the most user friendly and easily available material that meets your requirements to make your process easier. Also consider the melt flow rate and drying requirements of the material.

Common Plastic Types:

- **Polypropylene (PP):** Lightweight, flexible, versatile and resistant to fatigue. The most user-friendly material overall.
- **Polyethylene (PE):** Extensively use in the packaging industry and available in a wide range of densities.
- **Thermoplastic Elastomer (TPE):** Used for gaskets, seals and other applications where elasticity and softness are desired.
- **Nylon:** High strength and wear resistance, ideal for mechanical components.
- **Acetal (POM):** Low friction and dimensional stability, good for gears and moving parts.
- **ABS:** Strong, impact-resistant, used for housings and enclosures.
- **PMMA/COC:** Transparent and more chemically resistant than polycarbonate.
- **Polycarbonate (PC):** Transparent and impact-resistant, used in safety applications.

Helpful Tips

FOR WORKING WITH DIFFERENT PLASTICS

Requires Drying Equipment	PP PE TPE	>	EASY During the learning curve, use these plastics only for rubber like parts (TPE), for flexible parts (PE) and harder parts (PP)
	Nylon	>	MEDIUM With a slightly different mold, you can use Nylon to make harder parts at HIGHER temperature but at NORMAL pressures
	Acetal (Delrin) & ABS	>	HARD - REQUIRES EXPERTISE These can be used at NORMAL temperature but at HIGHER pressures to make high strength parts with specially designed mold
	PC & Others	>	VERY HARD - REQUIRES EXPERTISE These can be used at HIGHER temperatures and at HIGHER pressures with specially designed mold just for the material

What is a Polymer?

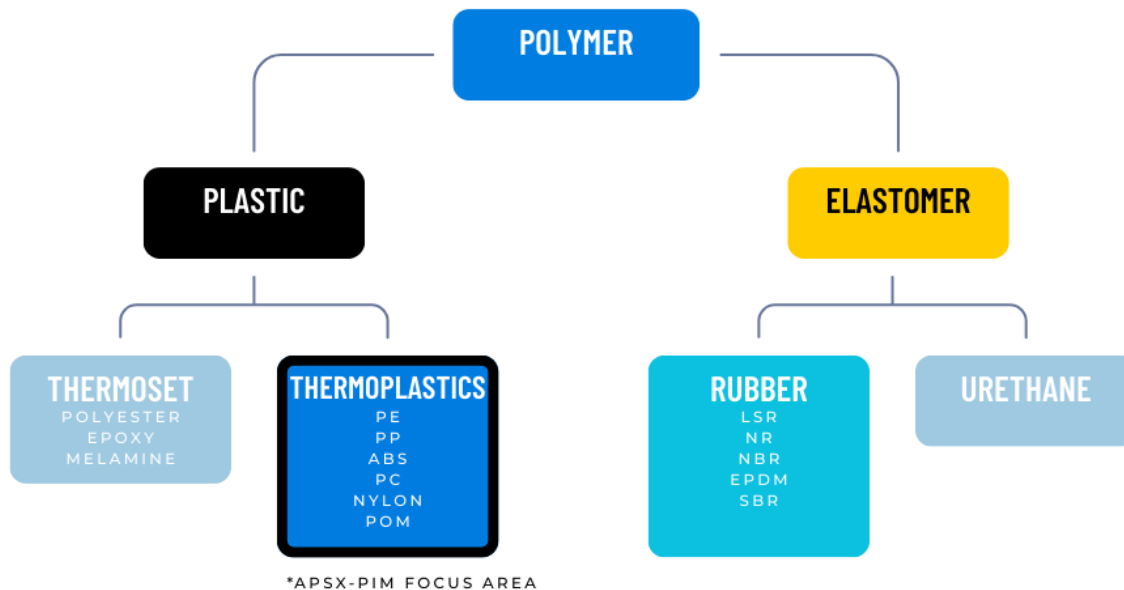
A polymer is a large molecule made up of repeating sub-units called monomers. These monomers are chemically bonded together to form long chains or networks. Types of polymers can have linear, branched, or cross-linked molecular structures.

Rigid Plastics: These are synthetic polymers that can be molded and shaped into various forms. They are typically derived from petrochemicals and can be further classified into thermoplastics and thermosetting plastics. Most linear and branched polymers are **thermoplastics**, meaning they soften when heated and harden upon cooling. In contrast, cross-linked polymers are **thermosets**, which irreversibly harden when exposed to heat or other initiating agents.

Elastomers: Elastomers are one of polymer types that exhibit rubber-like elasticity and flexibility. They can undergo large deformations and recover their original shape after the deformation force is removed. Rubber is a well-known example of an elastomer. Rubber materials are typically divided into natural rubber (NR), silicone rubber (SR) and synthetic rubber.

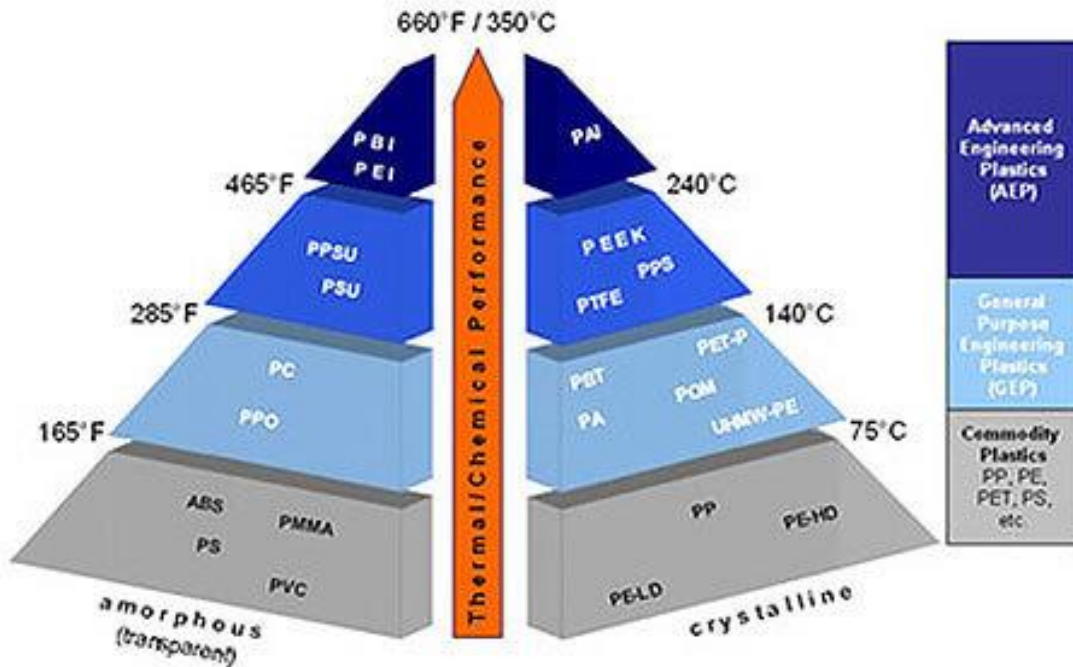


POLYMER CATEGORIES



Thermoplastic Materials

This is the **main type of plastic being used for injection molding**. They are linear polymeric materials that “soften” when heated and “resolidify” when cooled. They are also **recyclable**, however the material properties of a recycled polymer may not be equivalent to those of the virgin polymer.

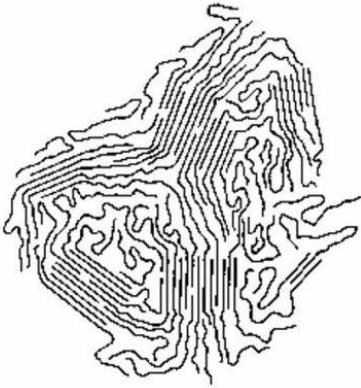


Amorphous Thermoplastics

These consist of polymer molecules with no particular conformation (random). When they are heated, the chains become more active, disengaged and the chain slippage occurs. They gradually soften and flow **with no sharp melting point**. **ABS, Polycarbonate (PC) and PEI (Ultem)** are examples for this type. They are often transparent, have good impact resistance and easy to form.

Semi-Crystalline Thermoplastics

Some polymers can form ordered rather than random molecular arrangements. Upon heating, the crystals remain intact until the polymer reaches its **distinct melting temperature**. Polymers such as **polyethylene (PE), polypropylene (PP), acetal, peek and nylons** are examples of semi-crystalline polymers. They are more rigid than amorphous ones, and have higher heat resistance.

Semi-crystallineAmorphous

General Plastic Characteristics

Understanding the properties of various plastics is fundamental to designing effective injection-molded parts.

Material Properties:

- **Mechanical Properties:** Tensile strength, impact resistance, and elasticity.
- **Thermal Properties:** Heat deflection temperature and thermal expansion.
- **Chemical Resistance:** Compatibility with environmental exposures.

Melt Flow & Viscosity

Different plastics have varying melt flow characteristics, affecting mold filling.

Important Factors:

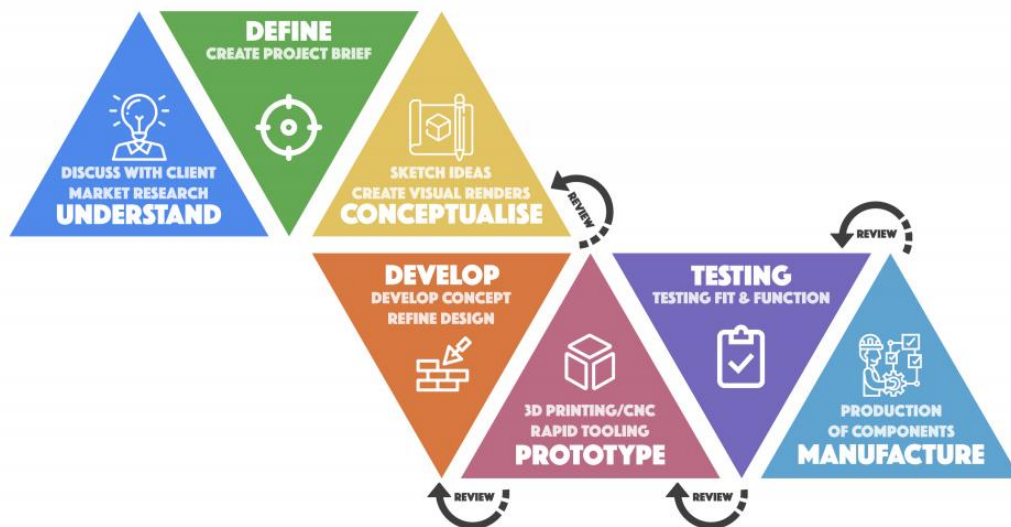
- **High Viscosity:** Requires higher injection pressure and longer fill times.
- **Low Viscosity:** Flows easily but can cause flash and excessive shrinkage.
- **Shear Rate Sensitivity:** Some materials degrade or burn at high shear rates, requiring careful process control.
- **Melt Flow Rate (MFR):** Indicates how easily plastic flows and influences process parameters.

4 – Fundamental Part Design Rules

The Design Process

An integrated approach to design and material selection leads to optimal performance. The process can include the marketing, prototyping, engineering, tooling and manufacturing teams. As a rule of thumb, the longer the process is, the more expensive the part will be.

PRODUCT DESIGN PROCESS



AFTER REVIEW, PROCESS CAN RETURN TO ANY PREVIOUS STAGE AS NEEDED

Product Design Process:

- **Define End-Use Requirements:** Anticipate structural loads, environmental conditions, dimensional requirements, regulations. Define mechanical and aesthetic needs, such as chemical and thermal resistance, cost, operating temperature range.
- **Prototyping and Iterating:** 3D printing of parts and molds, or a quick aluminum injection molds.
- **Testing:** Setup your production method once the prototyping result tests are successfully completed.

It is important to recognize that the design of the product will ultimately determine the “**ease of molding or manufacturability**” of the part, tooling requirements and the cost. Ideally, a plastic part should be designed so that it can be ejected from the mold without any special mold actions such as side action, and angle pins. Because such actions are expensive to design, manufacture and operate and add complexity to the process. In the following sections, we will review key part features that should be considered in the design process to simplify the design of molds.

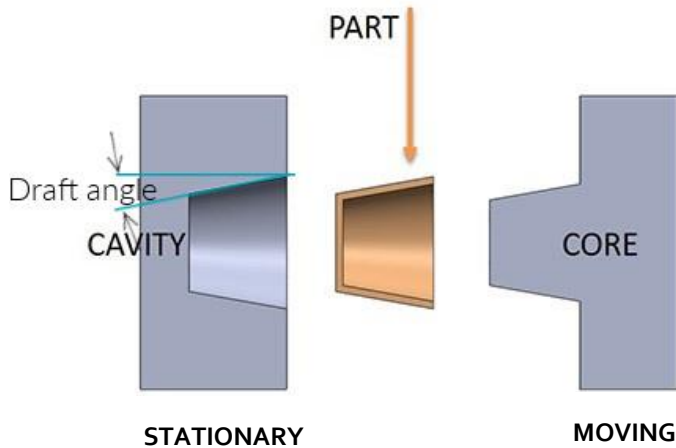
Wall Thickness

Thicker sections shrink unevenly, leading to warpage. Maintaining uniform thickness prevents warping and sink marks. Gating into the thick section is preferred because it enables packing of the thick section.

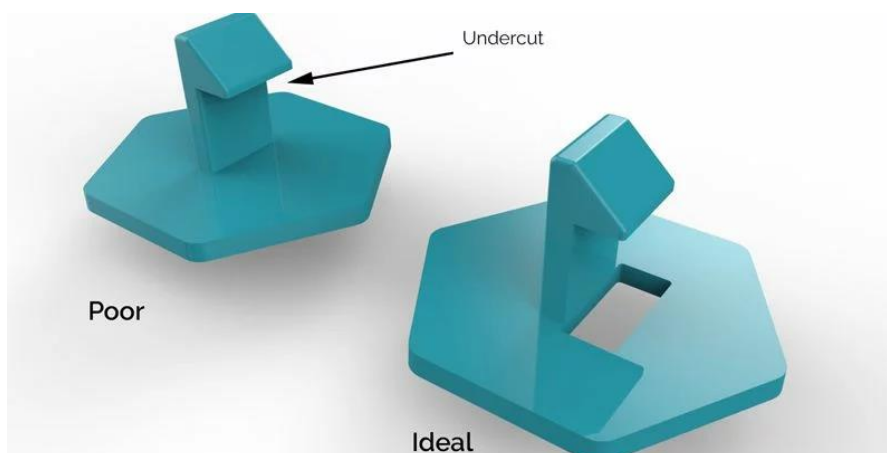
- Target 1–4 mm uniform walls.
- Core out thick sections; add ribs for stiffness.

Draft and Part Ejection

The plastic part must be designed with ejection considerations in mind. When parts are produced using the standard cavity and core configuration, the cavity is stationary while the core is attached to the moving platen. Parts tend to grip onto the core since the material shrinkage causes a contact pressure on the core. This tendency is considerably amplified where 3D printed molds are used, due to their surface and should be carefully considered in the mold design and the choice of 3D printer materials to be used for mold cores.



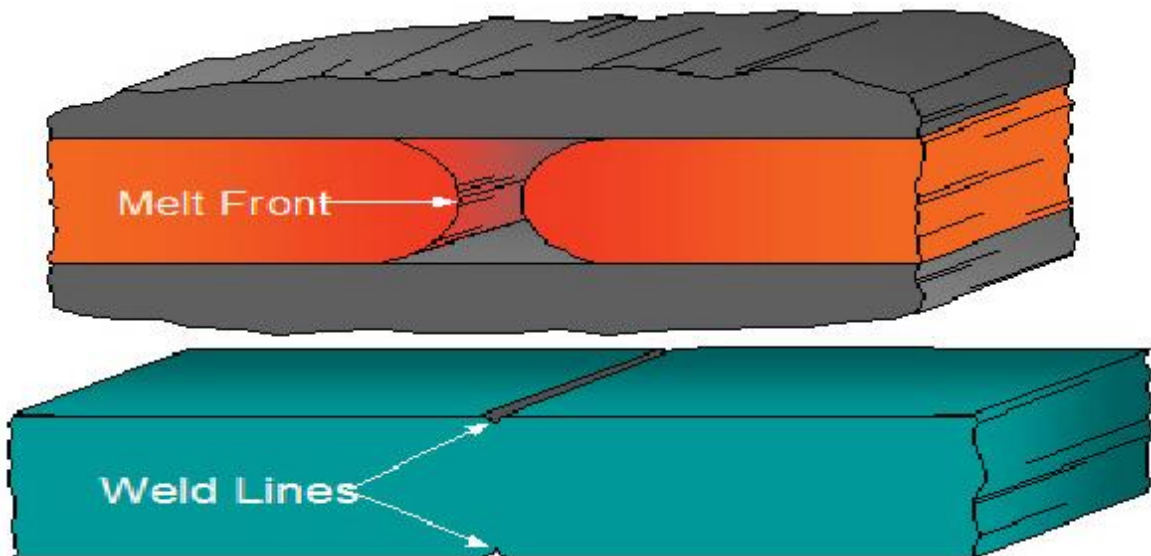
- For metal molds, draft angles from 0.5 to 2 degrees are commonly used. Increase 1° per 0.001" of texture depth. For 3D printed molds, 4 degrees or more is recommended. Alternatively, soluble mold materials may be used for mold features that risk getting locked into the molded part.
- Polishing a core or cavity in the line of ejection draw is preferred.
- It is also good practice to use bronze bushings to stabilize the ejector pins.
- Undercut internal cantilever snap. The use of a lifter can be eliminated by using a shut-off slot opening if esthetically acceptable.



Weld Lines

Weld lines look like cracks on the surface of the molded part. They form when two molten flow fronts meet, causing a potential weak point in the part structure. The problem is that the mechanical strength in the area of a weld line can be significantly lower than the other areas.

- Optimize gate location for shorter material flow paths.
- Increase mold temperatures to improve fusion.
- Use flow enhancers like additional vents.
- Try to avoid using materials with fillers such as glass or fiber.
- Add an overflow well adjacent to the weld to promote flow across the weld line.



Shrinkage and Warp

Plastic contracts as it cools, leading to dimensional changes that can cause defects.

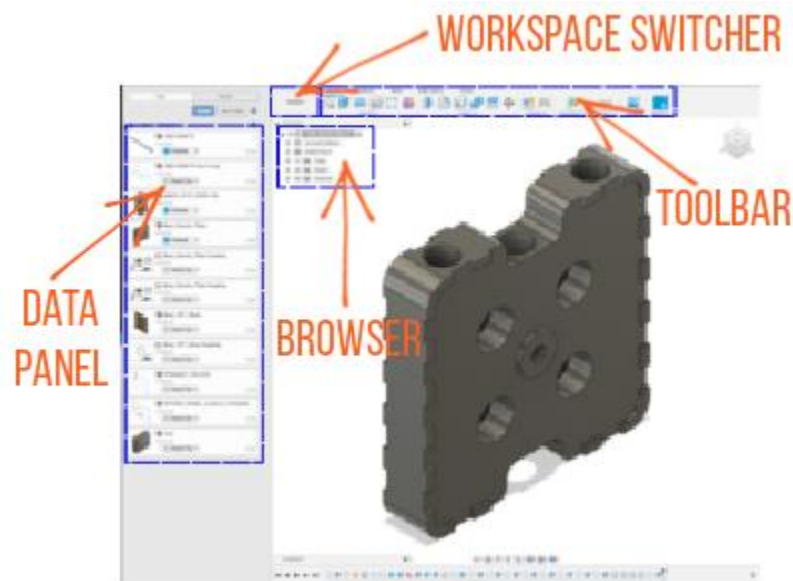


Material type: Amorphous plastics shrink less than semi-crystalline plastics. Common plastics such as ABS (0.5%), Nylon (1%), PP (2%) have different shrinkage percentages. Follow the manufacturer's suggested percentage when you design your mold for that specific plastic material. Where 3D printed molds are used, remember to factor in the shrinkage of the 3D printed material also (0.5-2%)

Holding pressure: Process variables such as the packing and holding pressures have a very significant effect on the shrinkage of a molded part. Holding pressure must be high enough to compensate for shrinkage effects, yet low enough to avoid an overpacking.

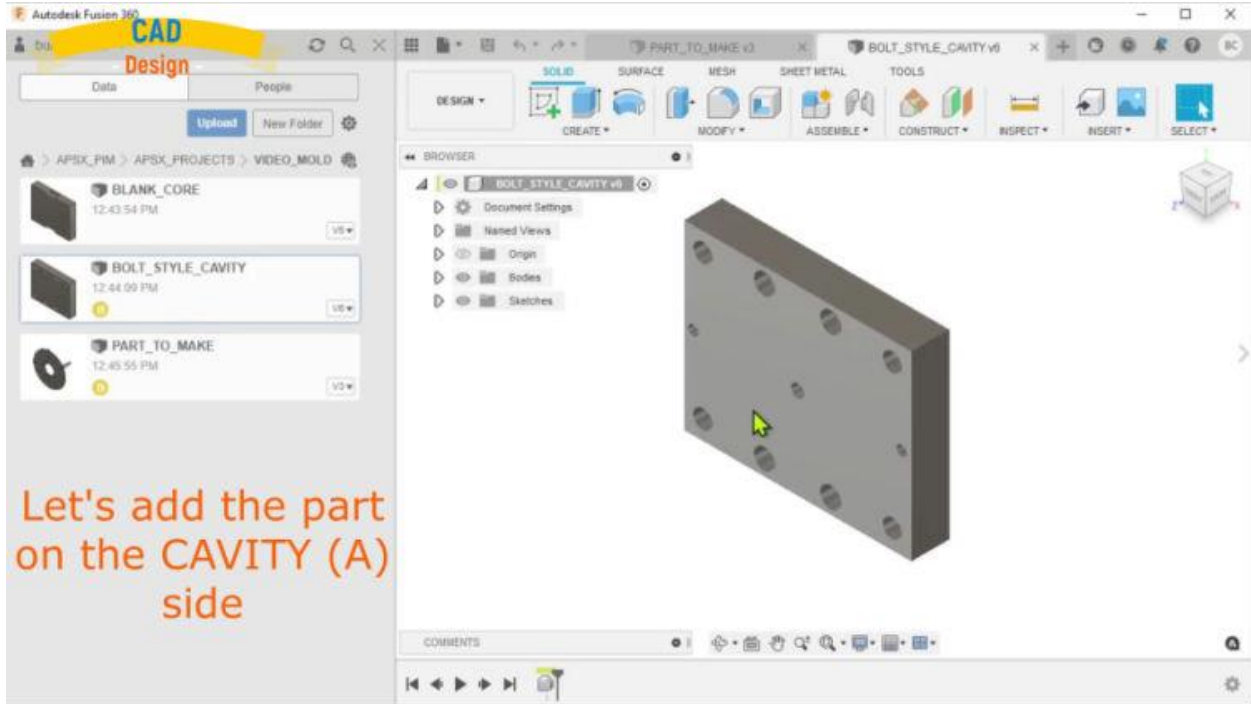
5 – CAD WORKFLOW (Fusion 360)

1 - The first step is to design the part in CAD software. The Fusion 360 is a powerful and also affordable option to use. But you can use any CAD software that you prefer.

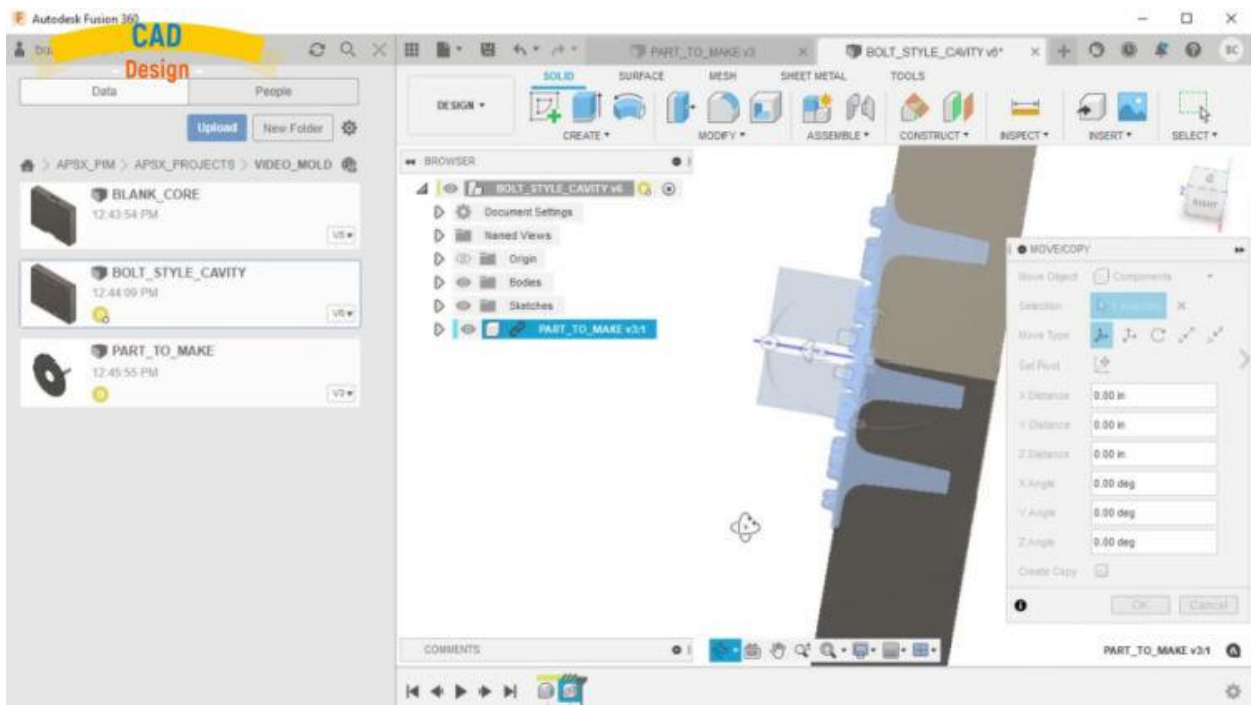


Once the part is designed in CAD, create a project for the mold you want to make.

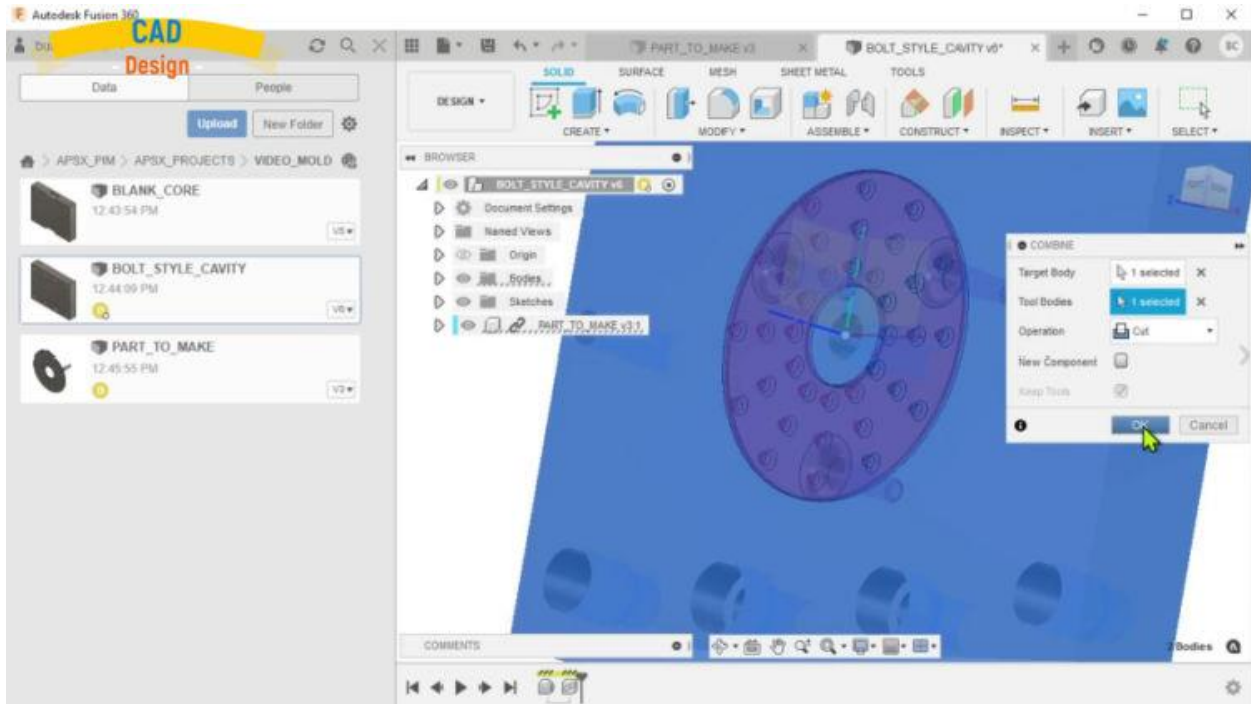
2 - Add the APSX blank mold template cavity and core files to your project.



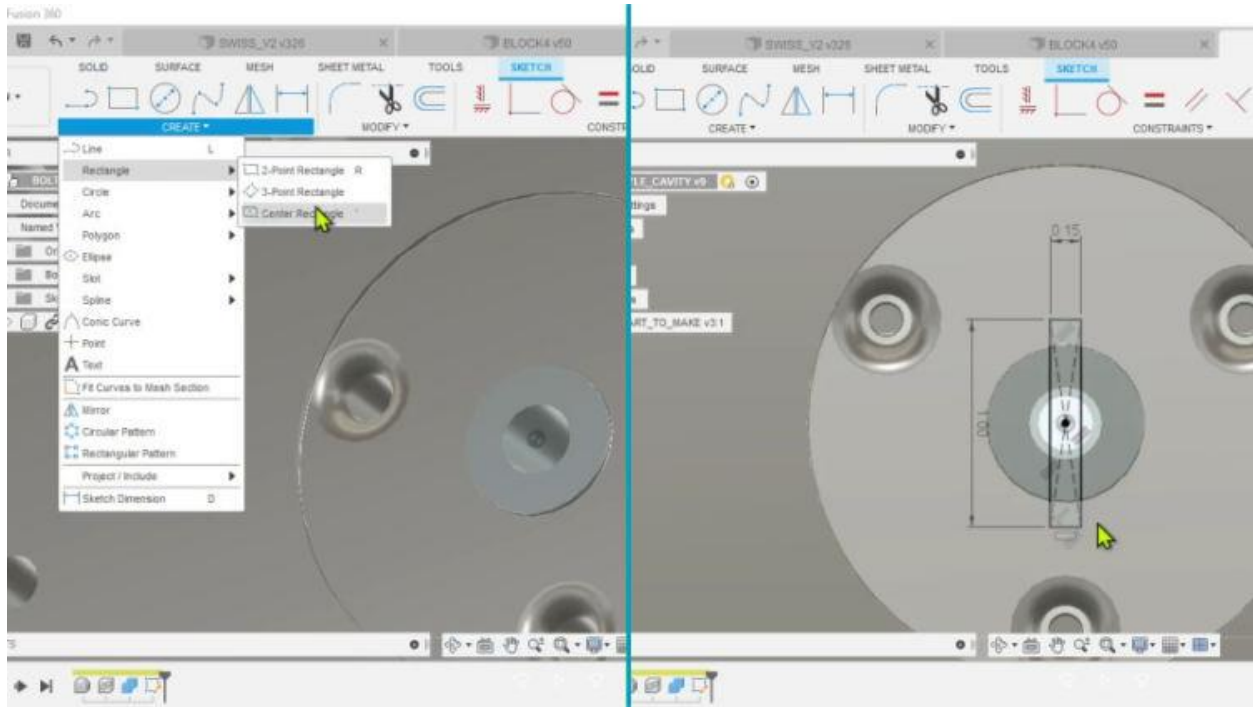
(POSITIONING) - Add the part to the cavity mold by clicking on the part file on the left data panel. Then adjust the orientation based on your part.



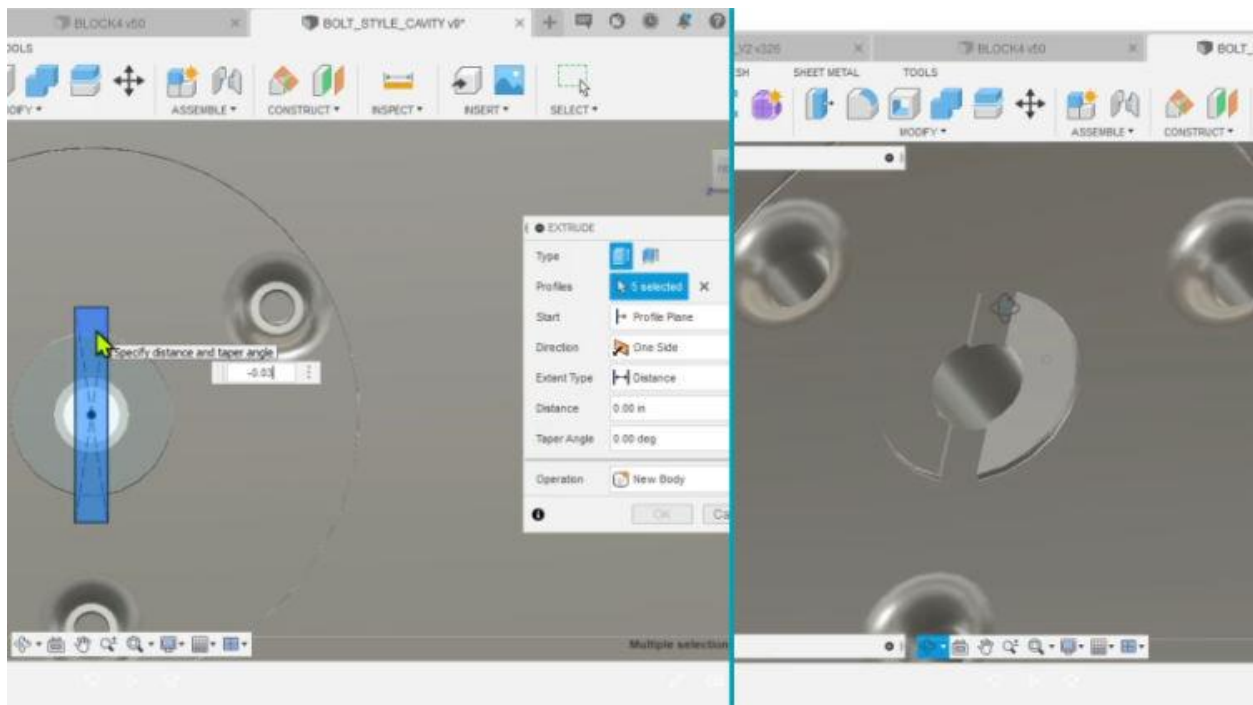
3 - Click on Modify menu on the top section and select the combine option. Select the mold and cavity as two bodies. Select the cut option for the operation to cut out the part from the mold.



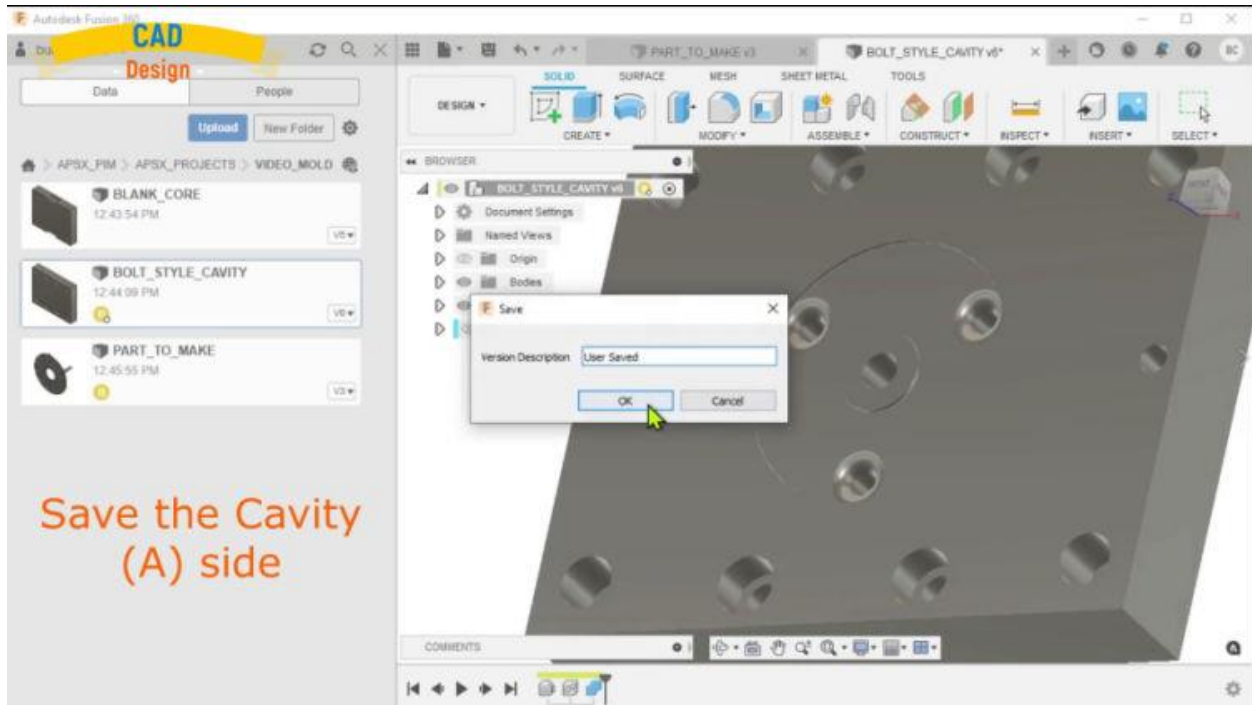
4 - Runner/Gate design: Select the center plane and select CREATE/Rectangle from the menu. Click on the center, hold and drag a rectangle. Adjust the dimensions as 1" by 0.15".



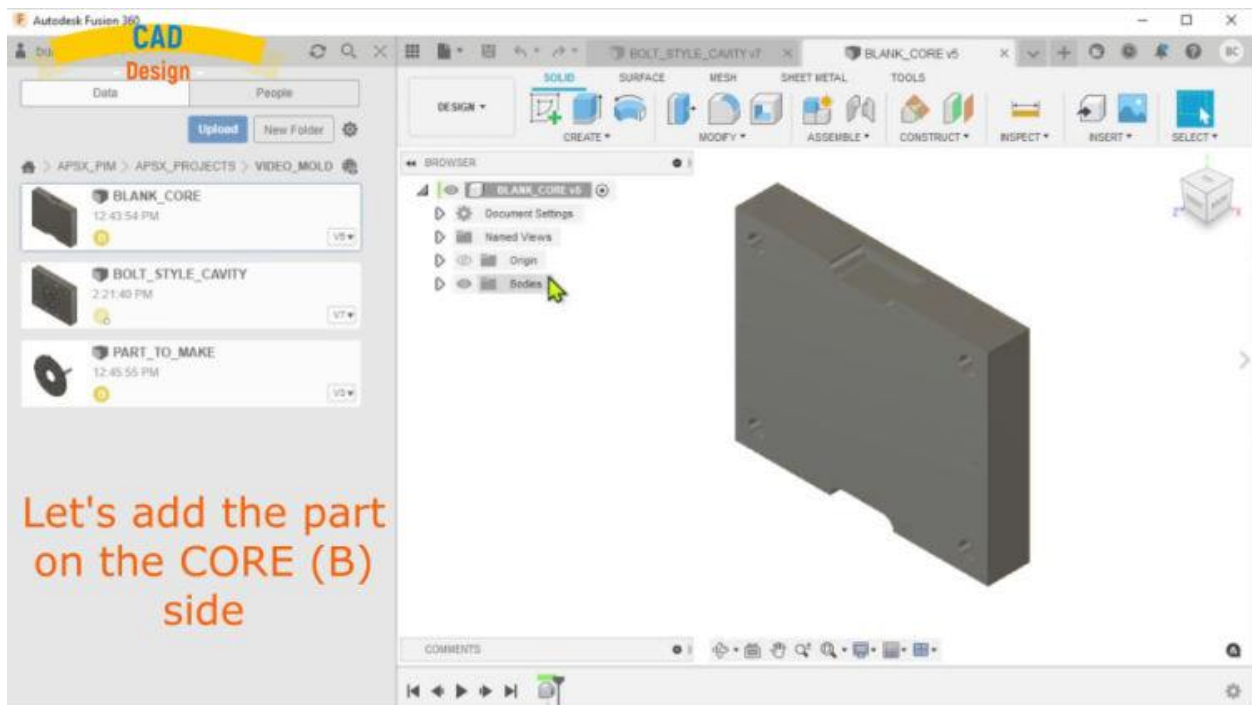
Select the rectangle and perform EXTRUDE (e) from the menu. Set the depth of extrusion as 0.03".



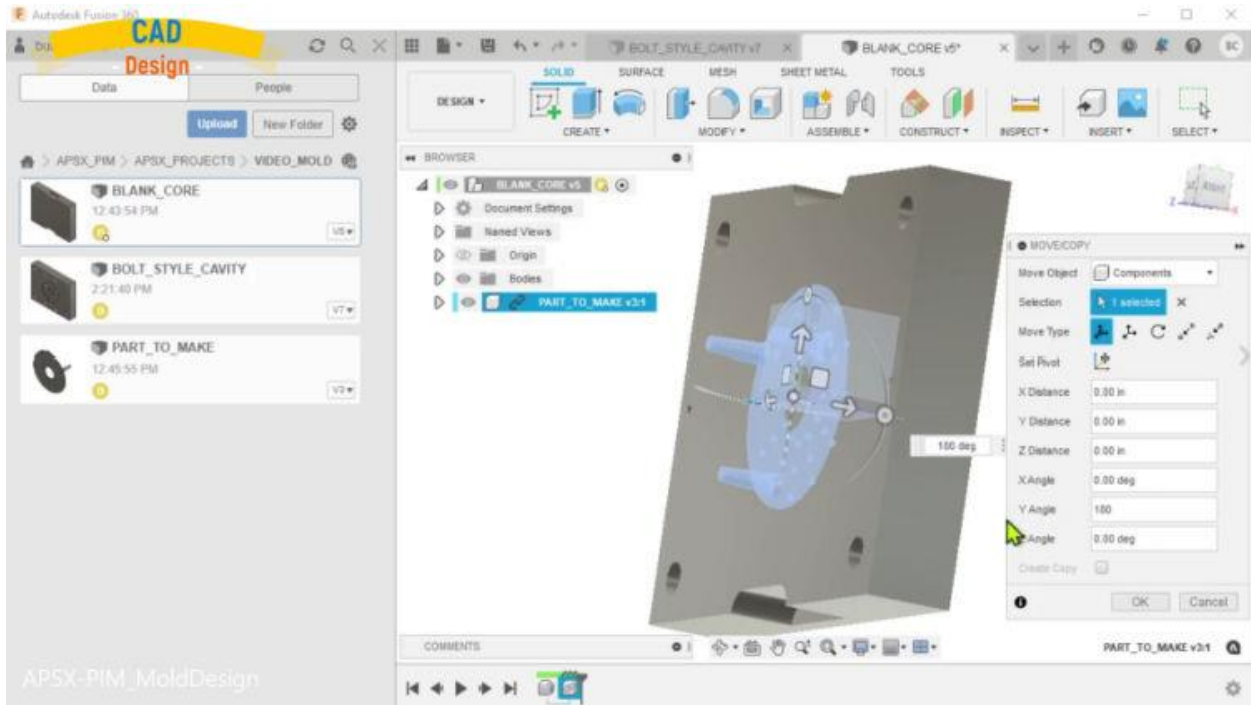
5 - You can add .001" deep venting lines if you need. Select the SAVE icon on the top menu to save the final mold cavity file.



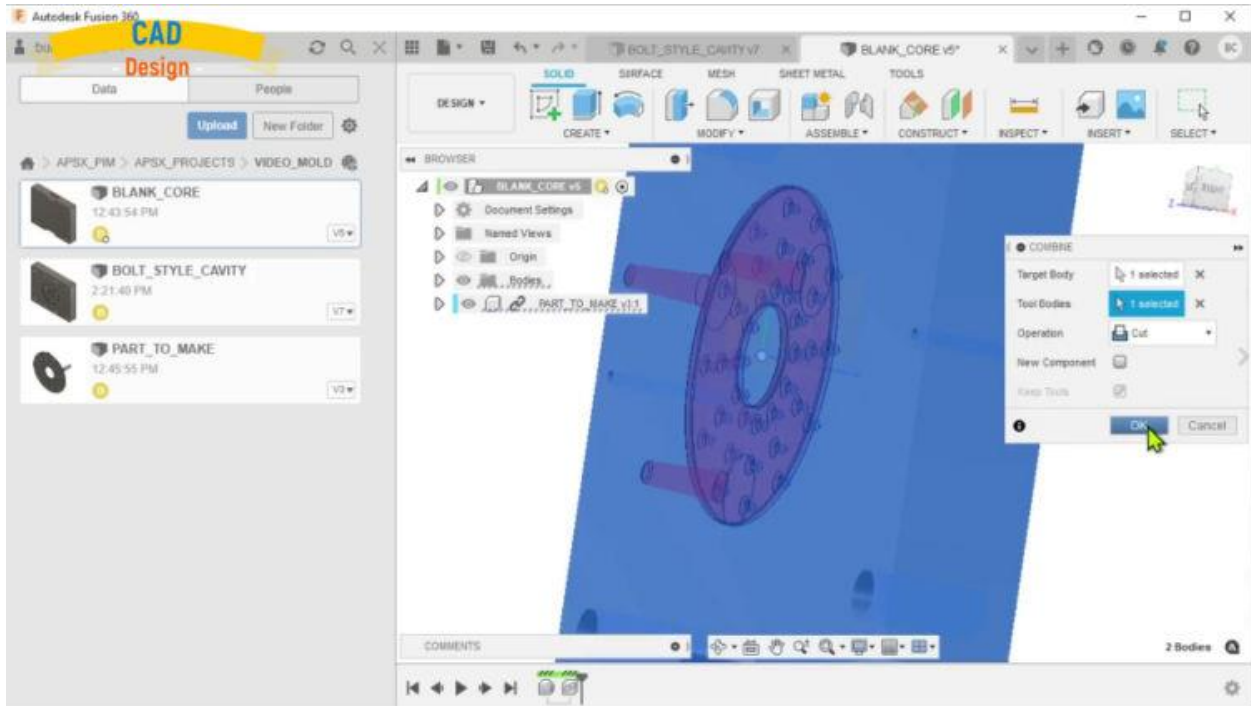
Now you can follow the same steps for the core (B) side. Click on the core side in the data panel to start working on it.



Add the part to the core mold by clicking on the part file on the left data panel. Then adjust the orientation based on your needs. You may need to flip the part 18- degrees.

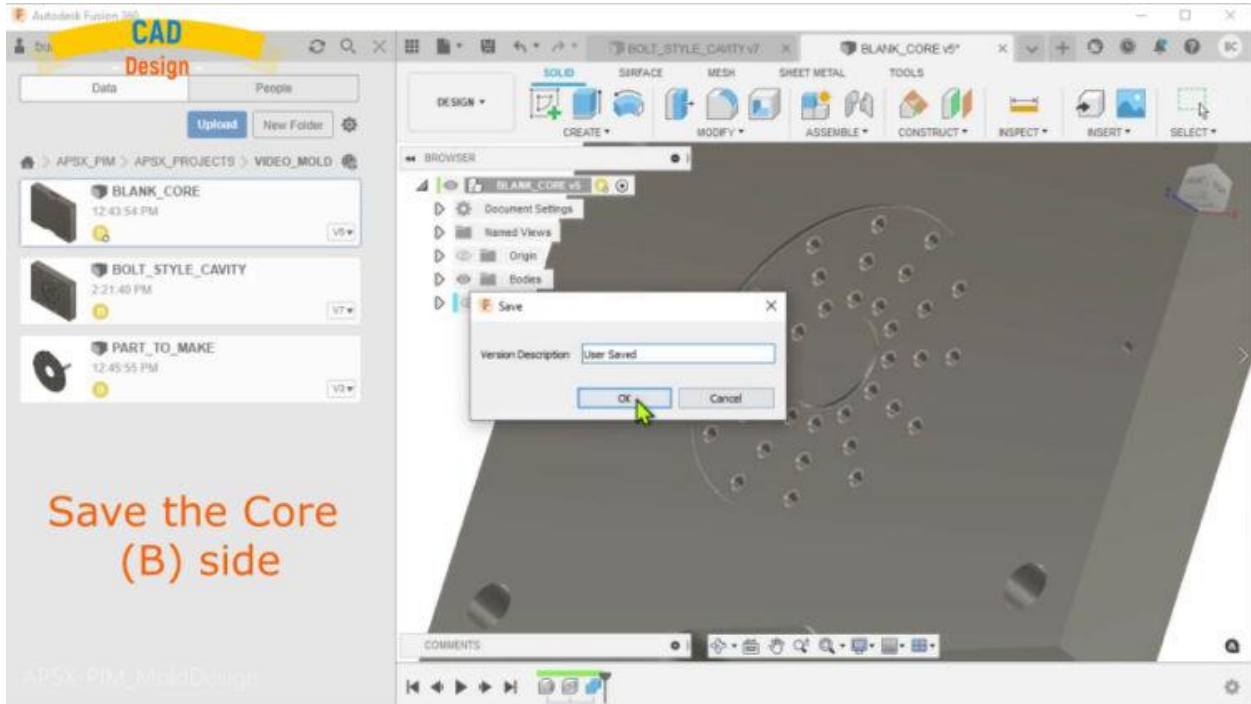


Click on Modify menu on the top section and select the combine option. Select the mold and core as two bodies. Select the cut option for the operation to cut out the part from the mold.



6 - Inspect the final mold core and make adjustments if you need. Select the SAVE icon on the top menu to save the final mold core file.

You are ready to move on to the CAM part of creating the machining paths and the g-code file for CNC milling the mold.



6 - Prototyping

Prototyping validates designs and identifies potential issues early.

Before full-scale production, prototyping helps validate design and functionality, reducing the risk of costly modifications.

Prototyping Methods:

- **3D Printing:** Fast and cost-effective for initial concept validation, but material properties may differ from final production plastics and the polymer type you want to use may not be available for 3D printing.
- **CNC Machining:** Produces accurate, functional prototypes from real plastic stock. Still the part performance may differ from that of molded parts.
- **3D Printed Molds:** Fast and cost-effective for initial part performance testing and mold concept validation.
- **Soft Tooling (Level I-II):** Uses low-cost molds for small production runs before committing to hardened steel tools.
- **Bridge Tooling (Level III):** Aluminum molds used for low-volume production before scaling up.

Benefits of Prototyping:

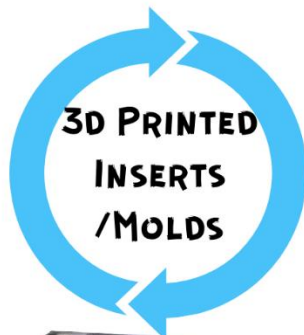
- Identifies design flaws before investing in expensive molds.
- Allows testing of part fit and function under real-world conditions.
- Enables iterative design improvements to optimize manufacturability.

7 - 3D Printed mold tips

While advances in simulations and CNC milling have dramatically simplified the manufacturing of prototype aluminum molds, the design and manufacturing these molds **for complex parts** in demanding materials remain a challenge. In addition, companies may not have the in-house skills to operate advanced CNC equipment, in which case mold manufacturing will have to be outsourced. Consequently, this process often becomes both expensive and time consuming, and may result in suboptimal designs as development teams skip critical iteration cycles to avoid excessive costs and delays.

The key to successful product development lies in achieving optimal design specifications before initiating production. Addressing imperfections during initial production stages typically incurs significant expenses. Recently, 3D-printed molds have emerged as valuable tools for accelerating iteration cycles while reducing costs, enabling teams to insource prototype mold making to finalize both part and mold designs more efficiently. In this way, 3D printed molds effectively bridge the gap between prototyping and full-scale manufacturing.

X NUMBER OF ITERATIONS




FINAL METAL MOLD DESIGN




Pros


3D PRINTED MOLD PROS



Much lower price:
Depending on the 3D printer you decide to buy, you can get started with 3D printed molds for about \$200. Compare that to an average of about \$10,000 for entry into industrial mold.



Immediate Implementation:
You can start right away since various suitable materials are now readily available for 3D printing applications. You need special 3D printing materials if you want to inject hot plastics.



Design Flexibility:
3D-printed molds offer greater design flexibility. If modifications are required after initial production, you can simply update your CAD model, implement the changes, and 3D print a revised mold.

Here are the biggest advantages:

- **Cost:** Much lower price point for rapid prototype molding and low-level batch manufacturing: Depending on the 3D printer you decide to buy, you can get started with 3D printed molds for about \$200. Compare that to an average of about \$10,000 for entry into industrial mold.
- **Immediate Implementation:** You can start right away since various suitable materials are now readily available for 3D printing applications. You need special 3D printing materials if you want to inject hot plastics. Examples include, the Rigid10K from Formlabs and the soluble mold resins from 3DPMolds.
- **Design Flexibility:** Traditional manufacturing mold production involves a complex, multi-stage process that limits modification once the final version is completed. In contrast, 3D-printed molds offer greater design freedom and

flexibility. Soluble mold inserts dramatically simplify the management of complex part features. If modifications are required after initial production, you can simply update your CAD model, implement the changes, and 3D print a revised mold.

Cons

3D PRINTED MOLD CONS



Not Ok for Large Batches:

If you're looking to directly jump into making batches of 500 or more, you're going to find your efforts in mold printing more laborious than helpful.



Design Limitations:

Basically, you can't just make anything because you have to be able to remove the mold after you've injected the material.



Wear and Tear:

A standard injection mold could be used to make hundreds of thousands of parts, but a 3D printed one can likely make hundreds of parts, depending on the materials used.

- Not suitable for large manufacturing batches: Although 3D printing molds make it possible to create a set of objects at the same time, there are size limits to that effectiveness. If you're looking to directly jump into making batches of 500 or more, you're going to find your efforts in mold printing more laborious than helpful unless you have the option of integrating 3D printed mold elements and conventional molding.
- Design limitations: Since your part has to come out of a mold, you lose some of the design freedom you have with 3D printing. Parts can't have encapsulated features. Basically, you can't just make anything because you have to be able to remove the mold after you've injected the material. However, the soluble 3D printing materials may be an alternative in prototyping process.

- **Wear and tear:** Since 3D printed molds are constructed of polymer materials that are less durable material than industrial molds, they're more likely to be damaged, especially when used regularly. A standard injection mold could be used to make hundreds of thousands of parts, but a 3D printed one will more likely last for ten to hundred parts, depending on the materials used and the design of the part.
- **Poor heat conductivity:** The heat conductivity of printed molds is roughly 1000 times worse than that of aluminum. Therefore, 3D printed molds will either have to be cooled after each shot or molds will have to be replaced between shots.
- **Part aesthetics:** The surface quality of a 3D printed mold will resemble that of a 3D printed part. While it is possible to polish and lacquer mold surfaces to increase smoothness, it is often more appropriate to consider implementing a metal insert in the 3D printed mold.

Part Design

Part selection is critical for setting yourself up to succeed as not all parts are great candidates for printed mold tooling. A good first step with part selection is to identify if a given part fits within your physical capabilities. If a part is “unmoldable” it doesn't matter what tools and techniques you implement during the design phase, you will be unsuccessful. If a part is too big for your frame or it requires a larger shot size than your machine is capable of, again you will be unsuccessful.

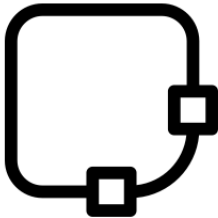
DFM (Design for Manufacturability) work includes: analyzing the material flow-path and optimal placement of gate and vents, adding draft to undrafted or underdrafted surfaces, breaking sharp corners with radii or chamfers, and in rare cases redesigning certain areas of parts for robustness. Along with this DFM work it is good practice to make note of “higher risk” features on the part. These part features include holes, ribs, and thin wall areas just to name a few.

DFM Practices



Draft Angle:

Adding draft to undrafted or underdrafted surfaces for easy removal of the final part.



Corner Radii:

Breaking sharp corners with radii or chamfers, and in rare cases redesigning certain areas of parts for robustness.



Higher Risk Features:

Note of “higher risk” features on the part. These part features include holes, ribs, and thin wall areas just to name a few.

At a minimum the 3D printed insert base should allow you to place a part in its optimal orientation, enable a robust ejection strategy, and leave 10+ mm of space between the part geometry and the outer walls of the mold block. Consider using 3D printed mold frames or inserts if you want to explore thinner mold walls or complex mold features.

Lastly, a scaling factor needs to be applied to the part itself. This scaling factor is used to combat plastic shrinkage, and it is a material-dependent value.

Material Selection

For most molding applications, consider using PP, HDPE, ABS, Acetal, TPU or TPE for an easier process.

For most molding applications,
consider using

PP

Versatile

HDPE

Tough

ABS

Impact-resistant

Acetal

Low-friction

TPU

Flexible

TPE

Elastic

MUD Base

Having a good template (master model) in place can save a designer multiple hours of design time on each mold. Not to mention the peace of mind that comes with knowing that all features are properly oriented and aligned. This master model is something that needs to be built once for each frame that you have in house. At APSX, we have a 3D printed mold template for 3D printed inserts and we can support with the design of 3D printed MUDS if you need something that is not available off the shelf.

Mold Design Considerations

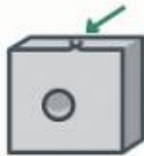
We hear all too often that someone tried 3D printed molds years ago and they just didn't work. And historically, 3D printed tools were not made from the right materials and/or printed on the right hardware to hold up in a mold press. While standard inputs like material properties and printer quality are a major focus, less obvious inputs like the **mold design**, mold material properties and the way 3D printed molds should be operated have turned out to be key contributors to successful molding outcomes.

3D Mold Design Considerations

To ensure success with printed molds, follow these rules of thumb and tips.

Draft Angles & Fillets

Allow $\geq 3^\circ$ draft angles and ≥ 0.1 mm radius fillets for part removal



Encapsulated Features

Avoid features that will trap the part in the mold

Easy release

Air Vent

Add vents to enable air escape from the cavity

Keying

Align mold halves correctly using pins and bushings



Parting Line

Plan around the line where mold halves join



Lifespan

Molds are made from a photopolymer, requiring different strategies

Design for a long life with hard mold

One very basic principle that has guided us is that 3D printed molds are fundamentally different from traditional steel molds. Therefore, we also need to approach mold designs differently.

There are some important design considerations to know about before you design your mold. Here are some rules of thumb and tips for getting the best results.

- **Draft angles and fillets:** The part design needs to have a draft angle on its faces so it can be easily removed from the mold. Allow at least 3 degrees of draft angle and 0.1mm or larger radius fillets to ensure the part will come out.

- **Air vents:** You need to include a way (0.1mm surface venting) for air to escape the cavity without letting the casting material spill out.
- **Keying:** Use keying to make sure the two halves of the mold always fit together correctly. A key is any shape that fits into another. Such as pins and bushings. Having one side of the mold keeping just flat is a method for simplicity.
- **Parting lines:** Two-part molds are the simplest, but you need to design around the center line that forms where the two halves of the mold join.
- **Lifespan:** The mold's lifespan will depend on materials and design and on the way the mold is operated. It is possible to operate a 3D printed mold like a standard metal mold, and attach it to the platen of the molding machine, but the machine-driven separation of the mold is stressful on the mold and part and may result in the mold quickly breaking down. To extend the life of a 3D printed mold, consider designing it so that it can be ejected with the part and taken apart outside the molding machine. Allow your 3D printed molds to cool down between forming cycles to extend their lifespan and prevent warping from repeated heating. That requires much longer cooling times than regular metal molds. Using larger gates is another factor of longer lifespan.
- **Complex part features:** Undercuts and part features cutting across the direction of pull may complicate the mold separation and part ejection.

Apply the same steps as in the standard mold design process mentioned for metal molds. Instead of using the standard mold blank, we need to use the 3D printed insert blank that goes into the metal MUD base that holds the insert in place.

3D PRINTED MOLD DESIGN




Use 3d Printed Insert Blanks

Use the 3D printed insert blank that goes into the metal MUD base.



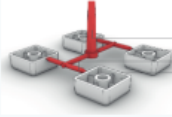
ADD



SUB

Create Core and Cavity Blocks

The cavity creation will be subtractive and the core creation will be additive.



Add Runner and Gate

Connecting the predefined runner start location to the target gate location.

Core and cavity blocks

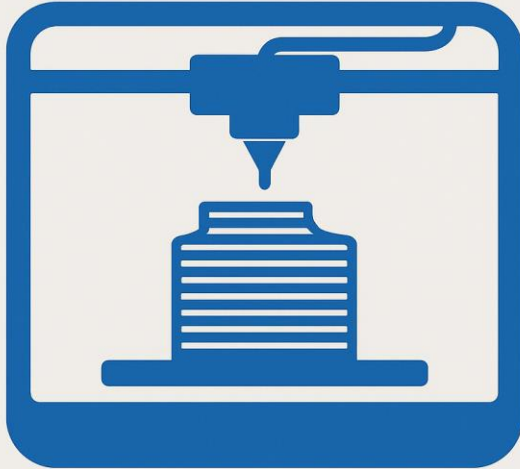
This requires both additive and subtractive CAD processes. Typically, the cavity creation will be subtractive and the core creation will be additive. Any recessed features in a mold block are created by merging the part with the 3D printed insert base and all standing features are created by copying the negative of the part geometry.

Runner & Gate

With the core and cavity created the last step is to add in a runner system and gate. This step is fairly straightforward as it is just connecting the predefined runner start location to the target gate location. This target gate location should be a flat face of the part somewhere on the parting line.

Apply a release agent before each forming cycle to facilitate easy separation of the part from the mold, especially when using flexible materials like TPE or TPU.

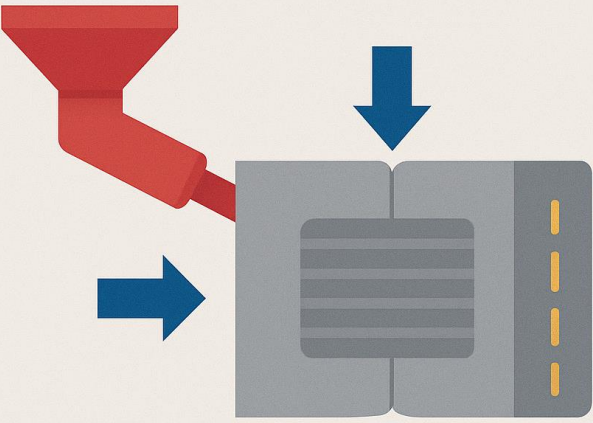
Print Settings



**PRINT AT THE
HIGHEST RESOLUTION
YOUR PRINTER CAN
MANAGE TO MINIMIZE
LAYER LINES
AND IMPROVE
SURFACE FINISH**




Print at the highest resolution your printer can manage to minimize layer lines and improve surface finish.

Injection Settings



MOLD COOLING

Use the lowest possible injection pressure and clamping force and allow longer cooling times for a proper mold cooling process.

-  Low Pressure
-  Low Clamping Force
-  Longer Cooling Time

Use fine-grit sandpaper for best results

Use the lowest possible injection pressure and the clamping force and also allow longer cooling times for a proper mold cooling process.

8 - BONUS

Mechanical Behavior of Plastic Materials

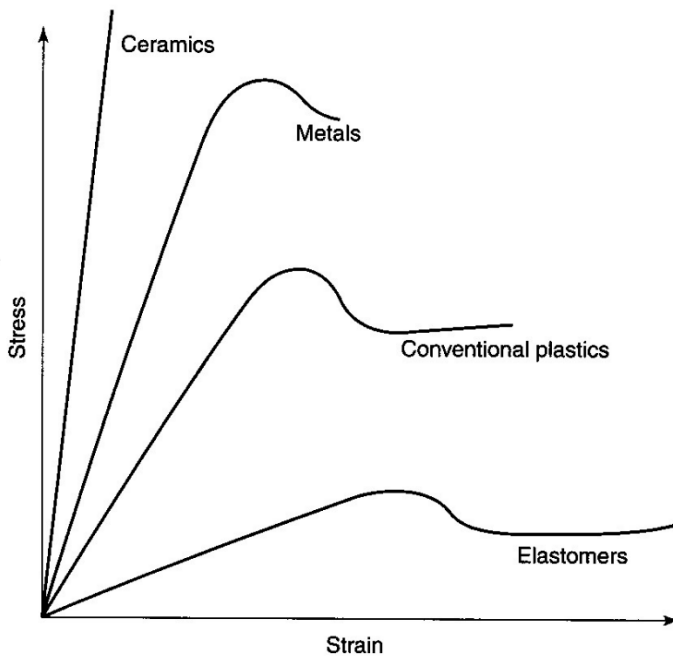
Material's mechanical properties are actually molded part properties rather than inherent material properties. There are factors including rate of loading, temperature, chemical environment, UV exposure, internal stress, and weld lines. The final part's mechanical performance can be tested by standardized test specimens such as dog bone specimens.

Stress-Strain (SHORT-TERM)

Plastic materials are less rigid (lower modulus values) compared to metals and woods. For example, a polycarbonate is only about $1/85^{\text{th}}$ as rigid as a carbon steel, and $1/28^{\text{th}}$ as rigid as aluminum.

Strain is typically given in units of 10^{-6} cm/cm for metals.

$$1000 \text{ ME} = 1000 \times 10^{-6} \text{ cm/cm} = 0.001 \text{ cm/cm} = 0.1\%$$



Yield Stress: The first stress value on a stress-strain curve where an increase in strain occurs without an increase in stress, if it exists. Examples: Titanium (900 MPa), A36 steel (350MPa), Nylon (45 MPa) and PP (30 MPa).

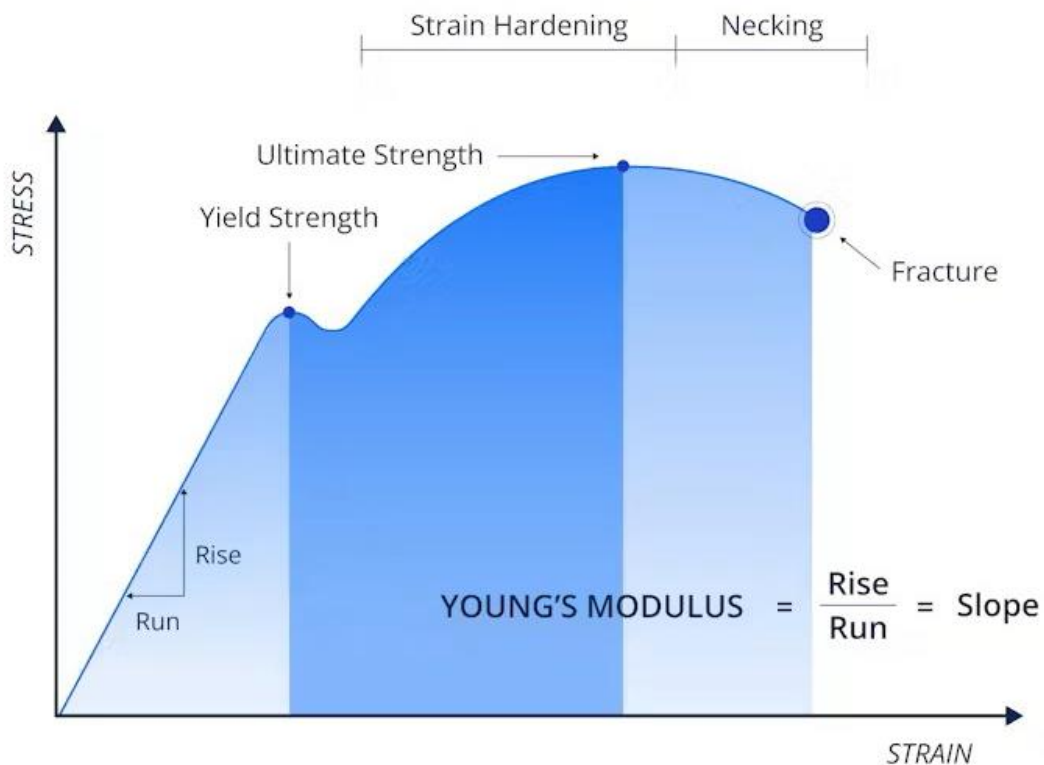
Yield Strain: The strain value (%) associated with the yield stress.

Toughness: The area under the stress-strain curve provides an indication of the overall toughness of the material.

Bending (flexural) Strength: It is a bending stress-strain data.

Shear Strength and Modulus: The shear modulus, G, of a polymer is the slope of the initial portion of a shear stress vs. shear strain curve.

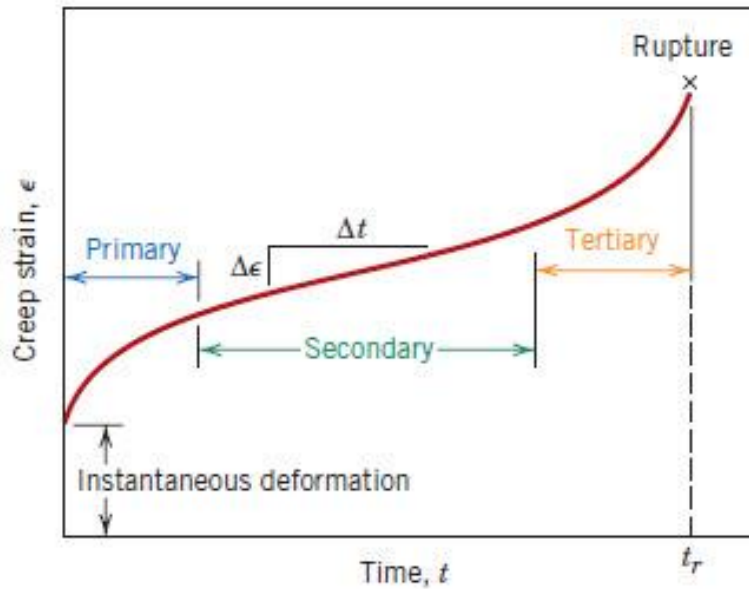
Stress Strain Curve



CREEP (LONG-TERM)

It is a long-term deformation concept in the long-term when the parts carry a load. When there is a load applied to the part for a certain time and removed, the part elongates

then recover back as much as possible but not 100%. Sometimes the part ruptures under the load stress.

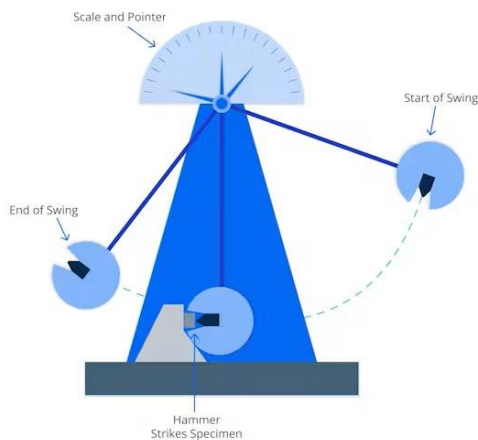


Impact Resistance (LONG-TERM)

Izod Impact

It measures the energy associated with failure of a notched cantilever specimen using a pendulum swing from a fixed height. In general, the impact strength of plastic materials increases as temperature and notch radius values increase. It is therefore important to use generous radius values on plastic parts to minimize stress concentration effects.

IZOD Impact Test



Tensile Impact

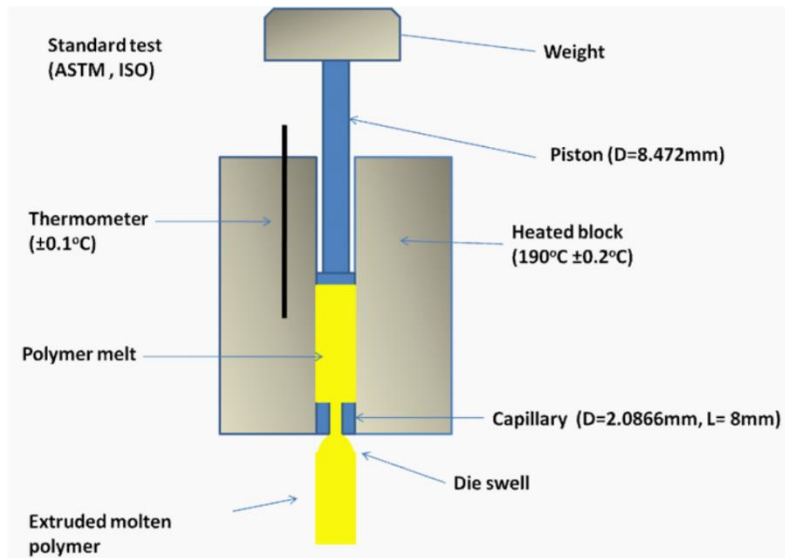
This test uses a small dog bone-like specimen that is subjected to uniaxial tensile loading. APSX offers different types of ASTM testing specimen molds.

Thermal Properties

Most plastics become brittle at some low temperature range and softer over some elevated temperatures. Amorphous materials such as polycarbonate does not have true melting point. It goes from glassy state to leathery state then rubbery then finally a liquid state in multiple steps. However semi-crystalline materials such as nylon or PP do have a true melting point. It changes from solid state to amorphous melt stage at that point.

Melt Flow Properties

The Melt Flow Rate Test (Melt Index) is the most common test to measure the flow property of the material. The flow rate (in units of grams/10 minutes) is measured at specific conditions of temperature and load. Low MFR means the material is tougher, higher heat resistance and overall better properties.



MFR does provide a rough indication of the processability of a polymer and does not describe the complete viscous flow behavior of the polymers. In fact, the flow rates associated with the MFR tests are orders of magnitude lower than those associated with mold filling.

IT IS POSSIBLE THAT TWO POLYMER GRADES HAVE SIMILAR MFR VALUES,
BUT VERY DIFFERENT MELT VISCOSITIES AT SHEAR RATES ASSOCIATED WITH
INJECTION MOLDING.

Glossary

Barrel: The section of a molding machine containing the feed screw or the plunger and the area where resin heating and mixing occurs.

Blushing: The tendency of a plastic article to turn white or chalky in highly stressed areas.

Bubbles: Air or gas pockets that have formed in the material of the component.

Cavity: A depression, or a set of matching recesses, in a plastics-forming mold that forms the outer surfaces of the molded articles.

Clamp: The part of an injection molding machine incorporating the platens that provide the force necessary to hold the mold closed during injection of the molten resin and open the mold to eject the molded part.

Clamping Force: The force applied to the mold to keep it closed, in opposition to the compressed molding material's fluid pressure within the mold cavity and the runner system.

Flow rate = $((\# \text{ of cavities}) \times (\text{volume per cavity})) / (\text{injection time})$

Cooling time: the elapsed time required for the melt to reach its Vicat softening temperature.

Core: A protrusion in a plastics forming mold that forms the inner surfaces of the molded articles.

Cycle Time: The time required by an injection molding system to mold a part and return to its original position/state.

Draft: A Slight taper in a mold wall designed to facilitate removing the molded object from the mold.

Drag Marks: A form of deep scratch or scratches on the surface of the component usually caused by the part's ejection.

Ejection Pin: A rod, pin, or sleeve pushes a molded part off a core or out of a mold's cavity.

Family mold: A mold that produces non-identical parts simultaneously from multiple cavities.

Flash: Any excess material formed and attached to the component along a seam or mold parting line.

Flow (Fill) Rate: the volume of material passing a fixed point per unit time.

Gate: The channel through which the molten resin flows from the runner into the cavity.

Injection Molding Pressure: The pressure applied to the cross-sectional area of the molding cylinder.

Injection Molding: The method of forming objects from granular or powdered plastics, most often of the thermoplastic type, in which the materials are fed from a hopper to a heated chamber in which it is softened a ram or screw forces the material into a mold. The pressure is maintained until the mass has hardened sufficiently for removal from the mold.

Insert Molding: Insert molding is the process of molding plastic around preformed metal inserts.

Machine Shot Capacity: Refers to the maximum volume of thermoplastic resin, which can be displaced or injected by the injection ram in a single stroke.

Material Safety Data Sheets: Documentation regarding the toxicity or hazards associated with contact with some substances. The manufacturer of the plastic prepares these datasheets.

Melt Flow Rate (MFR): A measure of the molten viscosity of a polymer determined by the weight of polymer extruded through an orifice under specified pressure and temperature conditions.

Mold Temperature: the temperature at which the mold is maintained.

Multi-Cavity Mold: A mold having two or more impressions for forming finished items in one cycle.

Packing: The filling of the mold cavity or cavities as full as possible without causing undue stress on the molds or causing the flash to appear on the finished parts.

Parting line: The marking on the part indicates where the two halves of the mold met.

Pellets: Tablets or granules of uniform size, consisting of resins or mixtures with compounding additives, prepared for molding operations by extrusion and chopping into short segments.

Platens: The mounting plates of a press on which the mold halves are attached.

Purging: In injection molding, the cleaning of one color or type of material from the machine by forcing it out with the new color or material to be used in subsequent production.

Runner: In an injection mold, the feed channel, usually of circular cross-section, connects the sprue with the cavity gate. The term is also used for the plastic piece formed in this channel.

Shrinkage Allowance: The dimensional allowance must be made in molds to compensate for the shrinkage of the plastic compound on cooling.

Sink Mark: an indentation on the part's surface due to significant local change in the wall section. The mark will occur in the thicker area.

Sprue: The feed opening provided in injection molding between the nozzle and cavity or runner system.

Tie-Bar Spacing: The space between the horizontal tie-bars on an injection molding machine. This measurement limits molds' size that can be placed between the tie-bars and into the molding machine.

Tool: In injection molding, the term is sometimes used to describe the mold.

Undercut: A protuberance or indentation that impedes withdrawal from a two-piece rigid mold.

Vent: A shallow channel or opening cut in the cavity to allow air or gases to escape as the melt fills the cavity.

Virgin Material: Any plastic compound or resin that has not been subjected to use or processing other than that required for its original manufacture.

Warpage: Distortion caused by nonuniform internal stresses.

Weld Line: Melted material flows together during molding to form a visible line or lines on a finished part that may cause weakening or break the component.



Contact Information



Address:

11121 Kenwood Road
Blue Ash, OH 45242

Phone:

513-716-5992

Email:

info@apsx.com

www.apsx.com