Accepted Manuscript. Published in the International Journal of Mechanical Engineering Education (IJMEE). This manuscript is made available according to SAGE's Green Open Access policy (<u>https://us.sagepub.com/en-us/nam/journal-author-archiving-policies-and-re-use</u>) under the following conditions: This manuscript is available free of charge, it is for non-commercial use, and derivative uses are not allowed. The citation for the final published pdf, which is available through IJMEE is:

Bradley RK. Education in plastics manuf

Creating an Aluminum Injection Mold

Robert Kelley Bradley

Department of Industrial & Systems Engineering

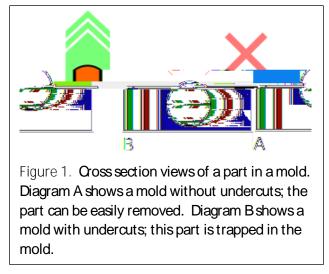
GUIDE

1. CAD Software:

1.1. Design the part

Once you are comfortable with CAD modeling, it is reasonable to use the CAD software to brainstorm a design. If you are new to CAD, it is best to start with a pencil and paper sketch. Once you have worked out all the details, you can then build it with the CAD software.

There are a few things that you should keep in mind as you start designing:



1.1.1 Avoid Undercuts

The part must be able to be removed from the mold. An undercut occurs when part of the mold sticks

out over the mold cavity, see Figure 1. Undercuts must be avoided because once the plastic part forms it will be locked in the mold.

1.1.2 Use Tapered Sides

It is always best to design your part so that it tapers in the mold, see Figure 2. A taper will allow the part to come free of the mold as soon as it moves upward. Even when an undercut does not exist, a part with vertical sides may still be difficult to demold, i.e. remove from the mold, because the friction force must be overcome as the part slides out. If the part has a vertical side that is less than about 1/8inch (3.175mm), you probably demolding. For example, if you wanted to make a coin

need to taper the sides of the letters.

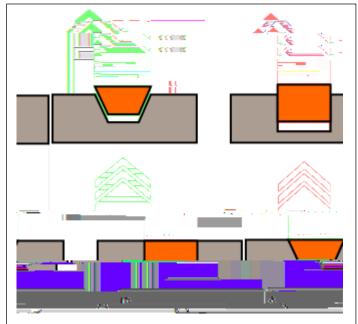


Figure 2. Cross section view of parts in molds. Diagram A has a tapered part that easily demolds. Diagram B does not have a taper and the part is difficult to remove due to friction.

1.1.3 Keep the Parting Line on a Single Plane

You will need to create a two-part mold in order to do injection molding. Usually both mold halves have cavities in them and when they fit together, the cavities create the 3-dimensinal shape of the mold.

common for a little plastic to push out between the two molds. You are probably familiar with seeing this thin line that runs around plastic called the parting line or mold line (and if there is extra plastic sticking out it is called flashing). Designing the mold is much easier if the parting line is completely contained within a single plane, see Figure 3.

1.1.4 Design

1.1.6 Consider the End Mill's Cross Section

End mill

A tapered end mill can be useful when making injection molds because, as was mentioned previously, we want to avoid straight vertical walls in the mold cavity. A taper can be created with a flat or round end mill, but a tapered end mill can do the same job much faster because it cuts a smooth taper regardless of the size of the stepdown. In most cases, the tapered end mill is

used in conjunction with a flat end mill. The flat end mill rough cuts the profile and the tapered end mill smooths it out, see Figure 6.

1.1.7 Parametric Modeling

Parametric modeling is a feature of some CAD programs, including Fusion360, which gives you an extensive capability to make changes to your model. As you enter dimensions and constraints, behind the scenes the CAD software creates a system of equations that describes you model. For example, if you have two concentric circles, where the inner circle is 2 inches (50.8mm) in diameter, you can specify that the outer circle should have a diameter 1 inch (25.4 mm) greater equaling 3 inches (76.2 mm); if

1.1.8 Designing on Paper

Though parametric modeling is very useful, and allows you to change

to start out on paper unless you are very comfortable with using the CAD software.

The point of designing on paper is not to make an

make a rough sketch and start adding dimensions. The goal is to make certain there are no major problems with your design. You will want to consider all the things that were discussed above (*e.g.*, undercuts, parting lines, end mill dimensions, etc.).

1.1.9 Example: Polyethylene Ball-and-Stick Model

As an example of utilizing the above ideas, we will look at the polyethylene ball-and-stick model from the accompanying paper. The idea is to create a single monomer with each mold so that they can be assembled to make a model of the polymer chain. The monomer will consist of a sphere representing

two pieces of the carbon atom stay together. Figure 7 shows the hand sketch and the ball detents used to solve the problem of holding the pieces together.

Figure 8 illustrates how two assembled carbon atoms can be attached together using a prong. This method requires that an aluminum rod be inserted in the mold so that when plastic is injected the rod will prevent it from filling the cavity. The rod must be removed once the plastic cools and hardens.

1.1.10 Create a CAD Model of the Plastic Part

The ultimate goal is to create a CAD model of the mold, but the easiest way to do that is to start by making a CAD model of the plastic part.

The primary way 3-dimensional

objects are create in Fusion360 starts with making a 2-dimensional drawing on a plane. The plane is often the XY-, XZ-, or YZ-plane, but can also be chosen

Creating an Aluminum Injection Mold

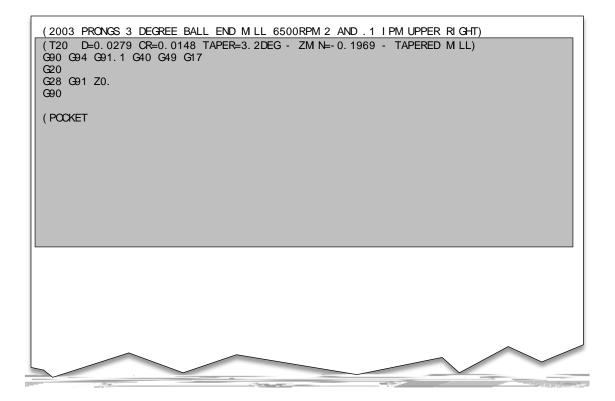


Figure 18. Illustration of how the G-code generated by the Fusion 360 post processor was modified at the beginning and end of the ASCII text file for use with the Mach III software that controls the TAIG ONC milling machine.

3. CNC Milling Machine:

3.1. Clamp an appropriate size block of aluminum on the CNC milling machine

The block of aluminum should be cut to the appropriate size. For the ball-and-stick polyethylene model, a 2 inch (50.8mm) x 3 inch (76.2 mm) x $\frac{1}{2}$ inch (12.7 mm) block should be used. The author used a chop

- 1) Carefully jog the end mill so that it just touches the left side of the block of aluminum near the upper left corner.
- 2) Zero the x-axis position in Mach III.
- 3) Carefully jog the end mill so that it just touches the top side of the block of aluminum near the upper left corner.
- 4) Zero the y-axis position in Mach III.
- 5) Carefully jog the end mill so that the tip of the end mill is well above the aluminum block.
- 6) If, for example you have a 1/8 inch (3.175 mm) diameter end mill, then in Mach III, in the MDI Line (Manual Data Input) type G0 X0.625 and enter. This will move the end mill 1/16 inch (1.5875 mm) to the right so that on the x-axis it will now be centered over the origin.
- 7) Next enter G0 Y-0.625 to move the end mill down by 1/16 inch (1.5875 mm) so that it is centered over the origin in the y-axis.
- 8) Re-zero the x-axis and y-axis in Mach III.
- 9) Carefully jog the end mill down until the tip of the end mill touches the surface of the aluminum block at the origin point.
- 10) Zero the z-axis in Mach III.

The author found the above method to be very adequate for purposes of making hobby injection molds, but an alternative approach involves using a center

3.6. Swap out different end-mills, re-zero, and repeat as needed

After running each G-code file, the end mill is replaced, the spindle pulleys adjusted if necessary, and the end mill position re-zeroed.

When multiple G-code files need to be run without replacing the end mill, the files can be combined by hand into a single G-code file.

4. Finishing the mold:

4.1. Polish the mold if necessary

After machining, the part is carefully deaned to remove leftover lubricant and aluminum chips. Machining marks and rough stepdowns can be removed by polishing with a rotary tool. The author used felt wheels and felt tips along with Dremel 421 Polishing Compound to smooth roughness in the mold cavities and remove machining marks. The results are visually attractive, but are not always necessary for hobby scale injection molding. In most cases the author found that polishing was not worth the time investment and simply used the molds as produced by the milling machine.

4.2. Insert alignment pins into one of the two mold halves