Bradley RK. Education in plastics manufacturing: Aluminum mold making and injection molding. *IJMEE*. November 2021. Copyright © 2021 the Author. doi:10.1177/03064190211051105

Creating an Aluminum Injection Mold

Robert Kelley Bradley

Department of Industrial & Systems Engineering

Lamar University, Beaumont, Texas 77705, United States

Supplemental Material submitted with

Education in Plastics Manufacturing: Aluminum Mold Making and Injection Molding

INTRODUCTION

Injection molding, even at the hobby scale, offers speed and precision in manufacturing plastic parts. Molds for hobby scale machines do not require the exacting detail of commercial molds, but creating one does entail many steps. Once you become familiar with the process it is relatively easy, but it has a steep learning curve. This document is a guide that is intended to help you learn the process.

This guide represents the knowledge gained from the author's experiences making injection molds over a period of several years. In writing the guide a balance was struck between providing step-by-step detail and general overview. In instances where the step-by-step detail was deemed directly relevant, it was provided. In other cases it is assumed that the reader can access the needed detail through help files and the internet. Though the guide is written based off experience with Fusion 360 CAD software and the Taig CNC mini mill, the information should be generally applicable to other CAD programs and other CNC milling machines.

The example mold that will be described can be used to make plastic parts that assemble into a ball-and-stick model of the polyethylene polymer.

SOFTWARE AND TOOLS

Fusion360 (Autodesk, San Rafael, CA) is used for both CAD modeling, CAM simulation, and post processing G-code. It is, as of this publication, is free to students, educators, and for personal non-commercial use.

The CNC milling machine described here is a Taig model 2000 HD-ER micro mill (Taig Tools, Chandler AZ) that uses Mach III software (Newfangled Solutions, Livermore Falls, ME) to control the motions of the mill. Mach III is often used for hobby scale CNC machines, but it requires an older style serial port that modern computers no longer have. It is therefore necessary to use a UC100 USB Motion Controller (CNC Drive, Pécs, Hungary), that can interface with a standard USB port.

Other useful tools include a chop saw for rough cutting aluminum blocks, adjustable wrenches, hex drivers, screw drivers, needle nose pliers, calipers, rulers, machinist squares/right angle gauge, a shop vacuum, a broom, etc. Proper PPE should also be used.

PERSONAL PROTECTIVE EQUIPMENT (PPE)

Safety glasses or goggles should be worn during all milling operations and for any other operations that could result in eye injury.

GUIDE

1. CAD Software:

1.1. Design the part

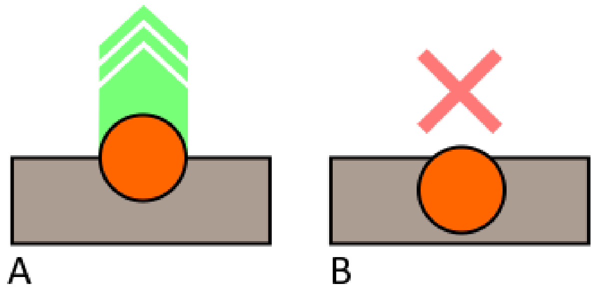

Figure 1. Cross section views of a part in a mold. Diagram A shows a mold without undercuts; the part can be easily removed. Diagram B shows a mold with undercuts; this part is trapped in the mold.

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

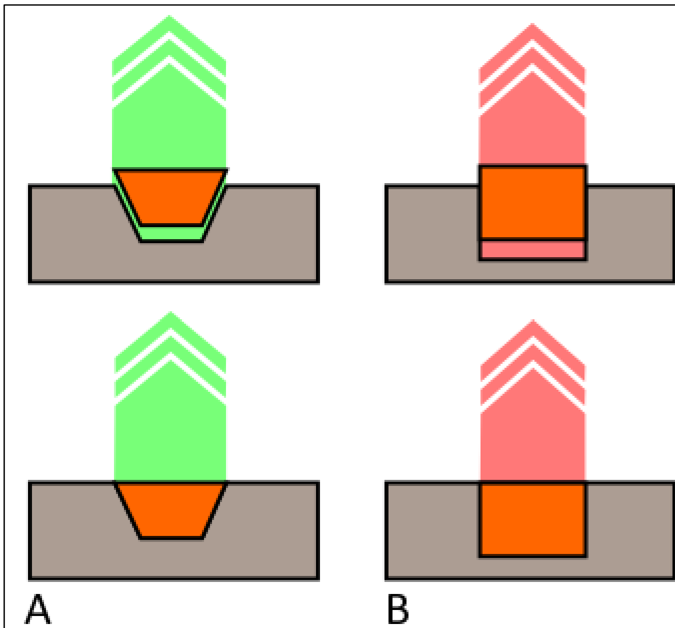

Figure 2. Cross section view of parts in molds. Diagram A has a tapered part that easily demolds. Diagram B does not have a taper and the part is difficult to remove due to friction.

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 won't have a problem demolding. For example, if you wanted to make a coin with letters on the face, you wouldn't need to taper the sides of the letters.

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. When injection molding, it's 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 parts; it's 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.

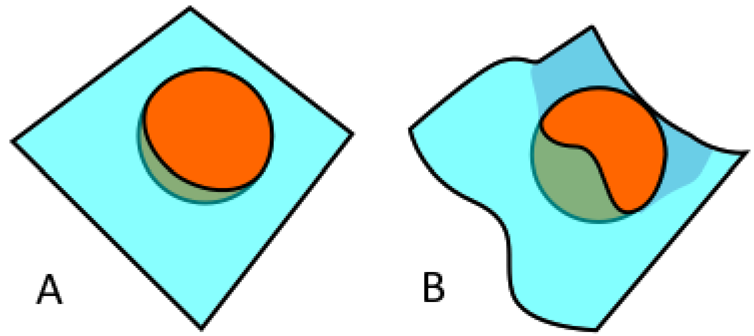

Figure 3. The parting line for two parts. In diagram A, the face of each mold half is represented by the blue plane. By keeping the parting line on a single plane, designing the mold will be much easier. In diagram B, the parting line is not on a single plane; the faces of the mold halves are not flat. It will be much more difficult to make this mold.

1.1.4 Design for Your End Mill Dimensions

With CAD software, you can create almost any object, however, just because you can create the CAD model doesn't mean you can actually machine the mold! It's very frustrating to spend a lot of time working on small details only to realize later that you don't have an end mill small enough to machine them out.

1.1.5 Plan for Inner Corners

Another common problem is machining internal corners into the face of the aluminum block. The end mill cuts out a circular profile, so it cannot cut square inner corners, see Figure 4. You can cut square outer corners, also see Figure 4.

Also, keep in mind that the end mill cuts the cavity in the mold, so when you think about your plastic part this will be reversed, you can get square inner corners but not outer corners.

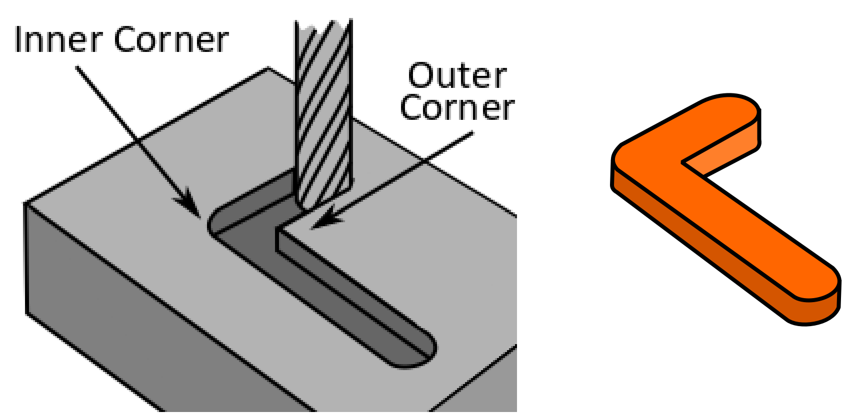

Figure 4. Internal corners. The end mill will cut its circular profile into the face of the aluminum block. Inner corners will be round, but outer corners can be square. Note that it is the mold cavity that is being machined so the actual plastic part will have round external corners and square internal corners.

1.1.6 Consider the End Mill's Cross Section

End mills come in a variety of shapes but there are three main ones that we are interested in: flat end mill, ball end mill, and tapered end mill.

In the previous section we saw how the axial profile creates round inner corners, but we also need to keep in mind the cross-section profile of the end mill when considering the cross section profile of the cavity that we want to cut.

Figure 5 illustrates how a ball end mill can give superior results over a flat end mill when the cross section of the cavity is curved. Of course, if you want a square profile, a flat end mill is preferred.

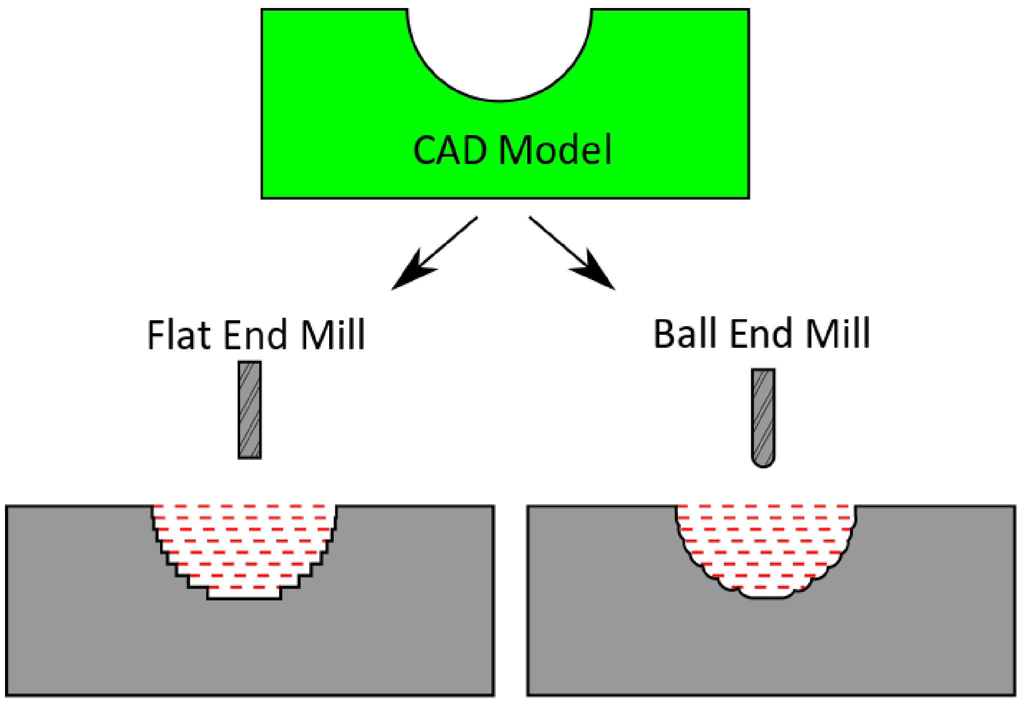

Figure 5. End mill cross sections. The CNC milling machine removes metal layer by layer. Using a regular flat end mill results in a "stair-step" profile, while using a ball end mill creates a smoother profile. By reducing the height of each stepdown, the ball end mill can create a very smooth contour.

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.

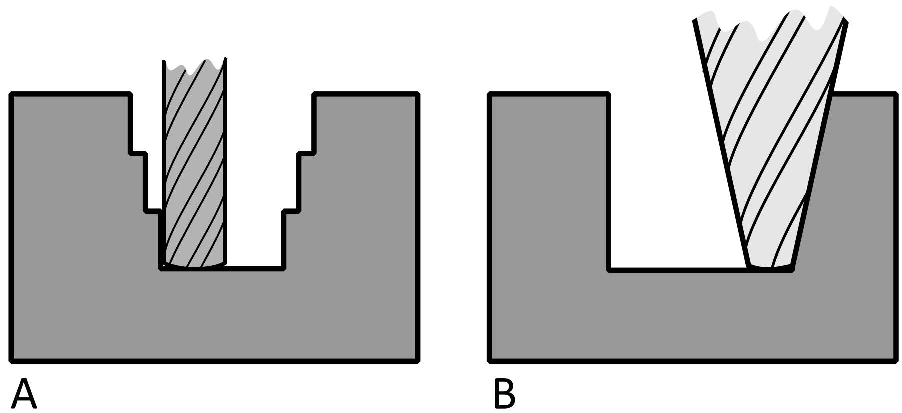

Figure 6. Milling a tapered profile. Diagram A shows how a flat end mill can create a stair-step taper by milling out wider and wider cross sections with each stepdown. Diagram B shows how a tapered end mill can make a smooth taper in one pass. Usually a flat end mill is used to create a vertical wall and the tapered end mill removes the remaining metal.

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 you later changed the diameter of the inner circle to 4 inches (101.6mm) the CAD software would automatically update the diameter of the outer circle to 5 inches (127mm). Once you get used to parametric modeling you will probably become a fan of it, but there are some aspects that can be annoying on occasion:

Constraints: A constraint is a way of defining your model without specifying dimensions. For example, a constraint may be that two lines should be perpendicular to one another. Fusion360 predicts what constraints you want and most of the time it’s right, however when it’s wrong it can be very vexing. If you try to move a line and it causes other parts of your model to change unexpectedly, it’s probably an issue with constraints.

Using Equations for Dimensions: in the example above, the outer circle’s dimensions were entered as an equation. If you just entered 3 inches (76.2mm), when you update the inner circle to be 5 inches (127mm) the outer circle would not change. You need to think about how a dimension should change relative to other dimensions and decide if you can simply enter a number or if you need an equation.

1.1.8 Designing on Paper

Though parametric modeling is very useful, and allows you to change your designs, it's still best 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 exact drawing, it's to 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.).

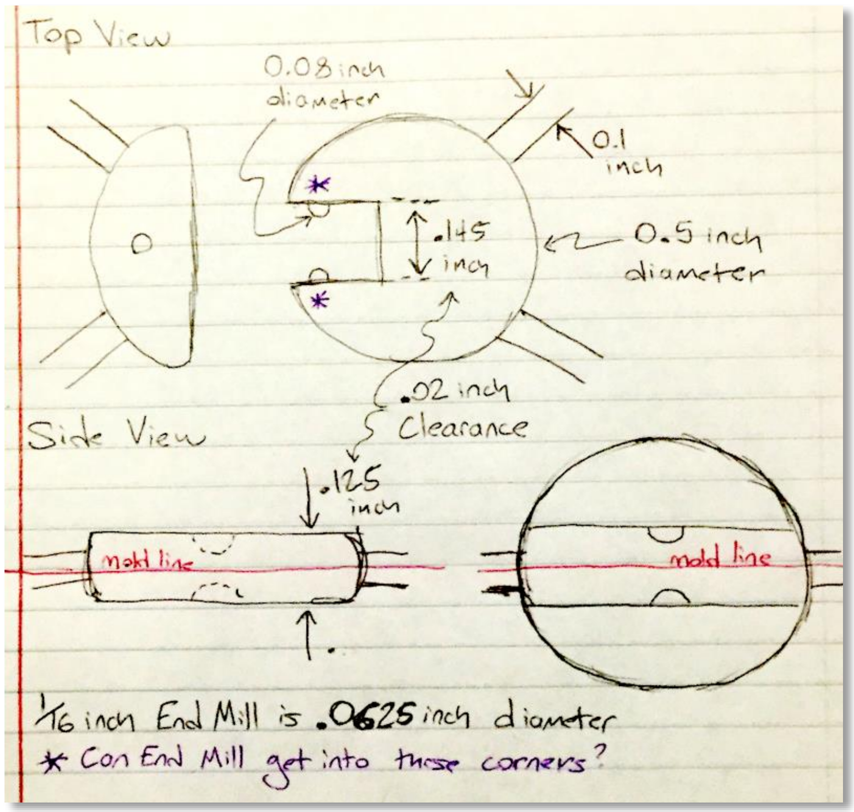

Figure 7. Paper sketch of the carbon atom in the polyethylene monomer ball-and-stick model. The sketch shows how the two pieces fit together. Two 0.08 inch (2.032 mm) diameter hemispheres inside the larger piece act as ball detents to lock the smaller piece in place. The sketch is used to make sure the dimensions can be machined with the end mill.

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 the carbon atom, two smaller spheres representing hydrogen atoms, and cylinders representing bonds. There are two initial challenges to consider:

1) How can the tetrahedral geometry of the sp^3 hybridization be created in a single mold?

2) How can the carbon atoms be attached to one another?

There are four bonds to the carbon atom and any two bonds are on a single plane. If the carbon atom were assembled from two parts, then each part could have two bonds on the same plane. This approach makes it possible to have a planar parting line, however it does raise the question of how the 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.

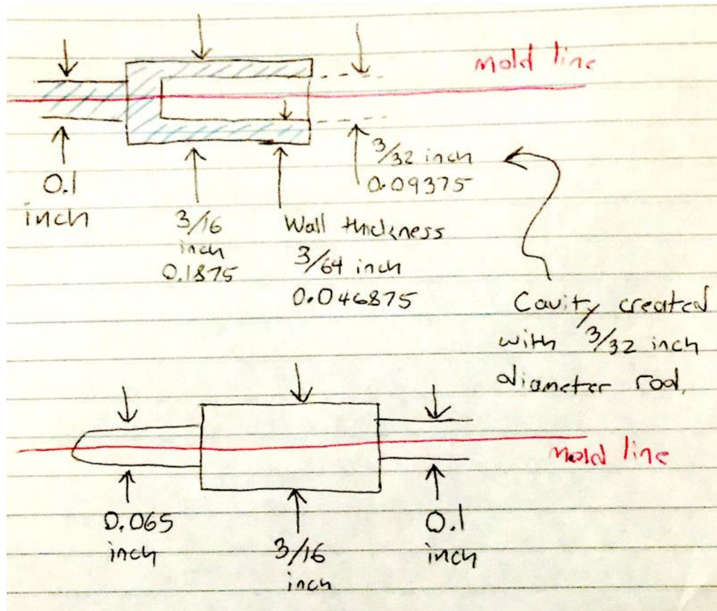

Figure 8. Two assembled carbon atoms can be attached together to model a covalent bond by inserting a prong on one bond into a cavity on the other. The cavity will be made by inserting a 3/32 inch (2.38 mm) diameter rod along the parting line between the two mold halves. The rod will effectively become a third part to the mold.

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 by selecting a plane on an existing 3-dimensional object or defined in other ways. Once a drawing is made on the plane it can be extruded or revolved to create a 3-dimensional object as shown in Figure 9. New objects can be independent objects, called bodies, or they can be added to or subtracted from an existing body to make a more complicated object, see Figure 10. There is usually more than one way to create a 3-dimensional model in Fusion360, but it's often the case that one way is easier so try to save time by planning your design strategy in advance.

1.1.11 Example: Polyethylene Ball-and-Stick Model

Figure 11 show shows the CAD model for the polyethylene ball-and-stick model. Figure 12 shows the initial 2D sketch for the piece that contains the two hydrogen atoms. The hydrogen atoms and their covalent bonds were made using a revolve. Initially the carbon atom was made using a revolve, but afterward Boolean subtraction was used to trim it into a 1/8" thick (3.175mm) slice and to add the depressions for the snap-together detents.

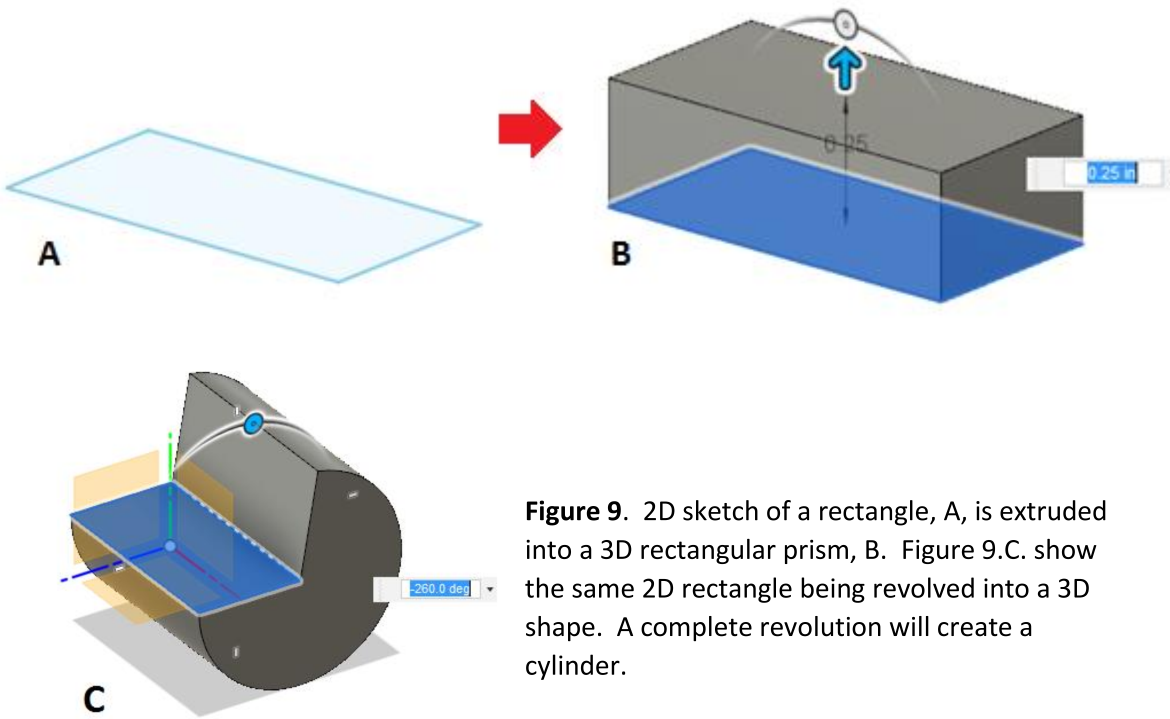

Figure 9. 2D sketch of a rectangle, A, is extruded into a 3D rectangular prism, B. Figure 9.C. show the same 2D rectangle being revolved into a 3D shape. A complete revolution will create a cylinder.

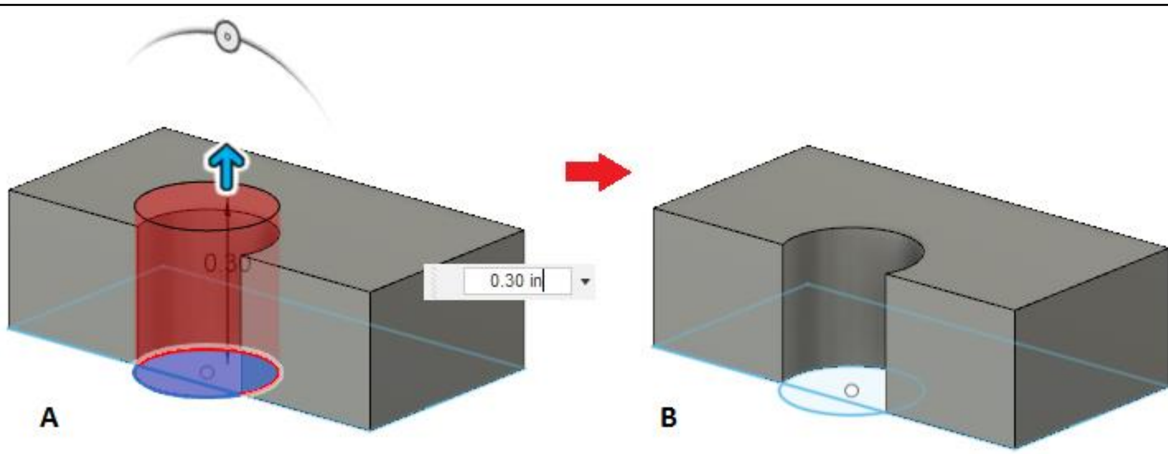

Figure 10. A cylinder, A, extruded from a circle is Boolean subtracted from a rectangular prism to make a more complicated shape, B.

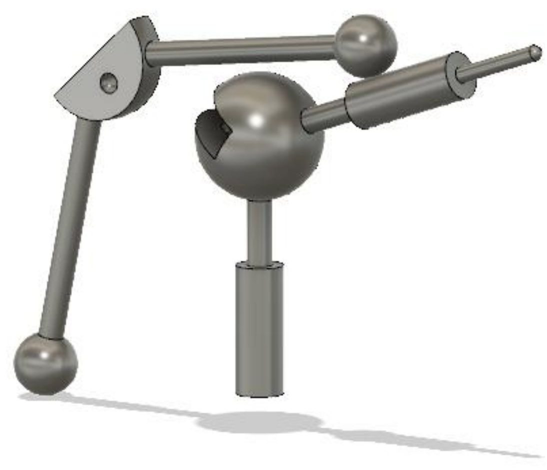

Figure 11. CAD model of the two pieces which snap together to make one polyethylene monomer. The initial sketch for the piece on the left is shown in Figure 12.

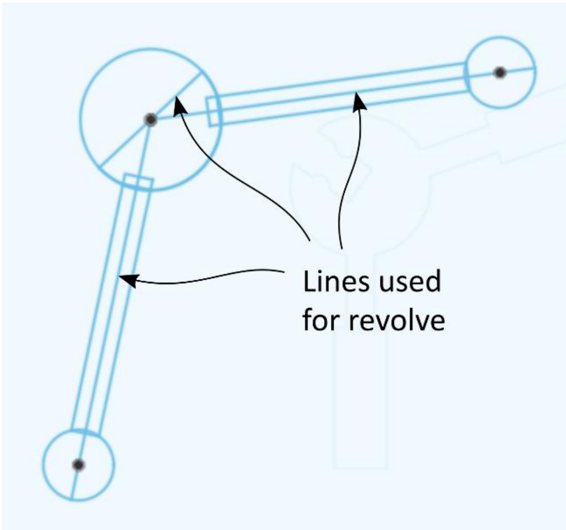

Figure 12. Sketch of piece with two hydrogen atoms. Revolve lines are shown. In the CAD file, dimensions are included, but they are not shown here for clarity.

1.2. Use Boolean subtraction to create the mold

First make a sketch of a rectangle that will define the plane for the parting line. The rectangle is extruded to make a 3D solid that is identical in size to the aluminum block that you will machine the

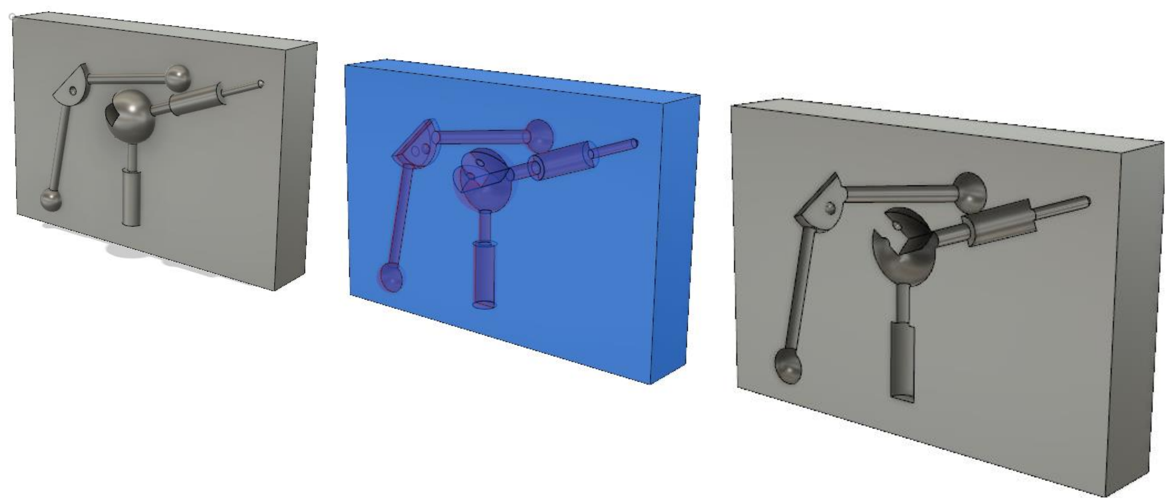

Figure 13. Creating the mold cavity with Boolean subtraction. On the far left, the ball-and-stick parts intersect with the block that will become the mold. In the middle, Boolean subtraction is being used to cut out the cavity. On the right the cavity can be seen.

mold from.

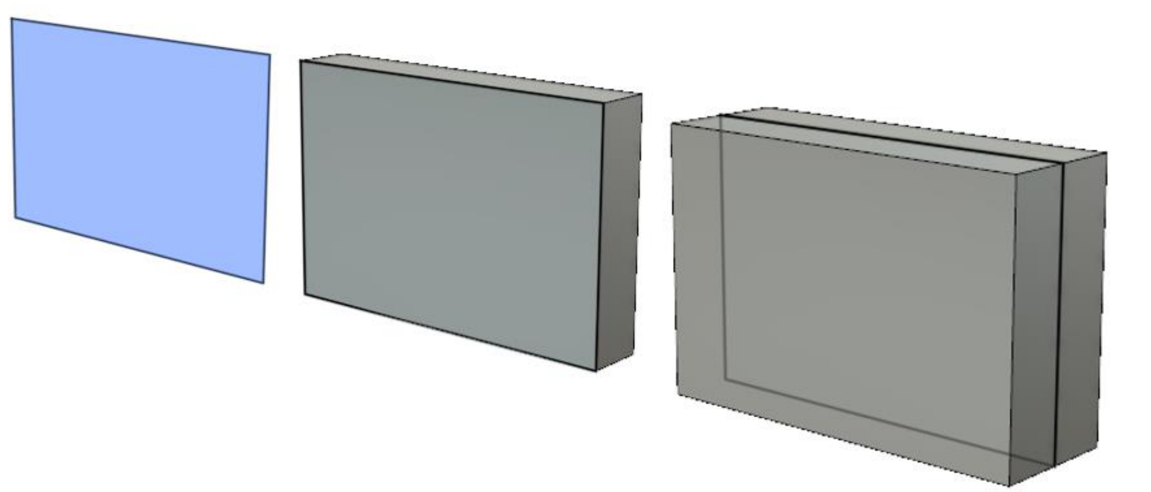

Figure 14. Creating the two mold halves. The sketch of a rectangle on the right defines a single plane for the parting line. The first mold half is created from the rectangle by extruding backwards. The second half is made by extruding forward. Both halves are perfectly aligned.

Figure 13 shows how Boolean subtraction is used to create the mold cavity. Fusion360 includes an option to save the model being subtracted so it can also be used to create the cavity in the second half of the mold. The same rectangle sketch is used to make the second half of the mold. When it is extruded, the extrusion is done in the opposite direction so that the two mold halves touch along the plane defined by the rectangle sketch, see Figure 14.

It's worth pointing out the need to keep everything aligned between the two mold halves, but it should happen automatically when following the procedure outlined here. Try to avoid modifying the mold cavities after the Boolean subtraction is done. If you need to make a modification, it's better to utilize the parametric modeling of Fusion360 to go back along the timeline in order to modify the original model. Fusion 360 will then make changes to all operations occurring later in the timeline, including the shape of the cavity that you created using Boolean subtraction. In this way the two mold halves will always be aligned.

1.3. Create the port, runners, gate and alignment pins as objects

The injection port is where the nozzle of the injection molding machine fits against your mold. If your injection molding machine allows you to place the port on the parting line, then it's a good idea to make a funnel shaped port for easier alignment with the nozzle.

Runners are channels in the mold that the hot plastic flows through during the injection process. They are mold cavities and will be filled with plastic, but they are usually cut off of the final part. Be sure to avoid undercuts when creating runners. A sprue is like a runner but is usually thicker and extends directly from the injection port, with smaller runners spreading out from the sprue. If the injection port is along the parting line, there may be little to distinguish a sprue from a runner.

To make removal of unwanted plastic from the part easier, a gate is placed at the point where the runner contacts the part cavity. The gate is simply a constriction of the runner so that plastic is pushed through a small opening. Once injection molding is complete, the gate creates a weak spot that is easily broken to remove the extra plastic.

Alignment pins make certain the two mold halves properly mate up. Unlike the port, runners, and gate, the alignment pin holes won't be filled with plastic, instead a pin is placed in them so that the two mold halves are guided together properly. The pins can be loose or they can be glued into one half of the mold with temperature resistant adhesive. The pins themselves can be made from aluminum rod cut to length.

1.4. Use Boolean subtraction to add corresponding cavities to the mold

As was done with the part to be injection molded, the first step is to create a model of the port, runners, gate and alignment pins. They will then be Boolean subtracted from the mold halves. Figure 15 illustrates this process. Note that for this particular mold, there is also a channel at the bottom to hold and align the 3/32 inch (2.38 mm) diameter rod described previously.

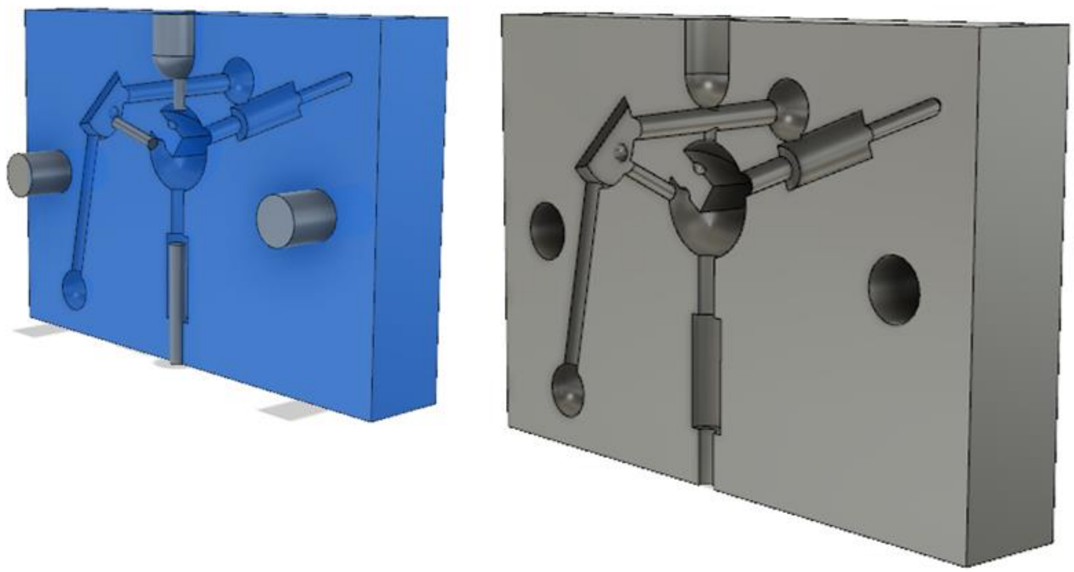

Figure 15. Boolean subtraction to make the port, runners, gates and alignment pins. The channel for the 3/32 inch (2.38 mm) diameter rod insert is also shown at the bottom of the mold.

2. CAM Software:

2.1. Enter data describing your end mills

Computer Aided Manufacturing (CAM) software can simulate the motion of an end mill as it mills away the unwanted material from a block of aluminum. These simulated motions are called toolpaths in Fusion360. In order to create a toolpath, data describing your end mill must be entered.

CAM features of Fusion360 are in the Manufacture workspace, and can be accessed using the change workspace button.

Fusion 360 comes with many end mills with preset data, however it's still likely that you will need to create your own. It's possible to create a local or even cloud based custom tool library, but another straightforward approach is to simply create a dummy project in which you save all your custom end mills. Later, all you have to do is open the dummy project along with your current project and Fusion360 will allow you to choose end mills from the dummy project for use in the current project. For fun, you can create a CAD model of a tool box in the dummy project to remind yourself that you are storing all your custom tool data there, see Figure 16.

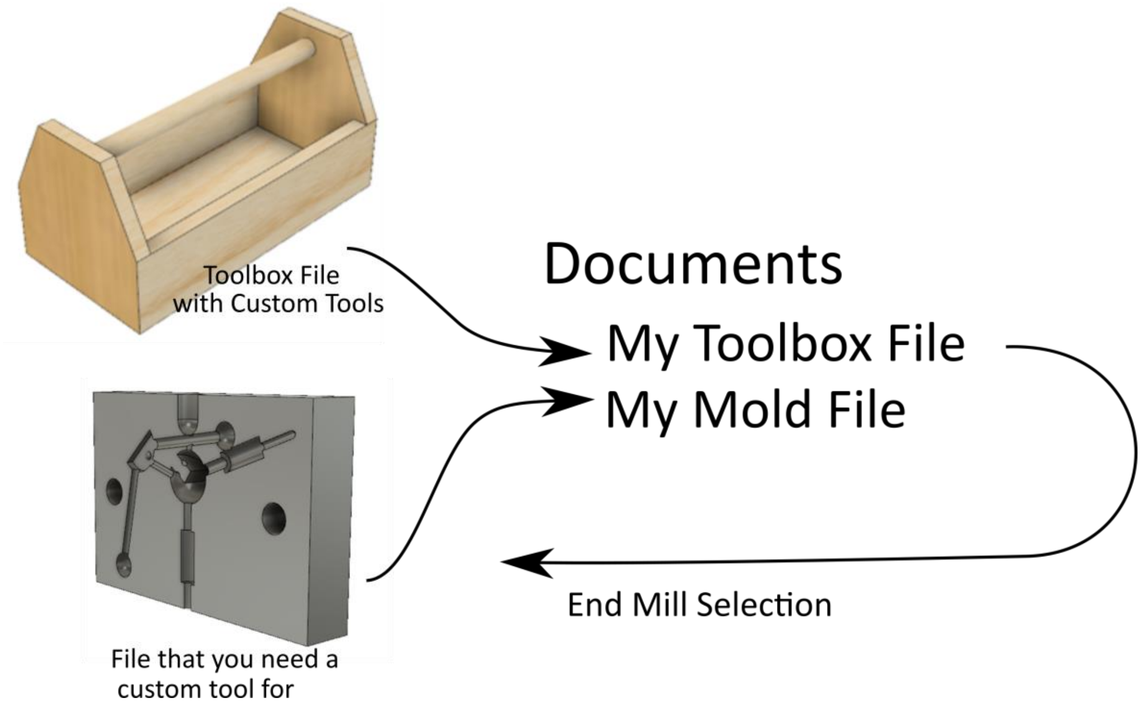

Figure 16. Two files are open in Fusion360, the toolbox file is where custom tools have been saved. The second file needs custom tool data prior to generating tool paths. Fusion360 allows custom tools to be selected from those saved in the toolbox file. This method may be easier than creating a local or cloud based custom tool library.

When you create a custom tool, Fusion360 allows you to enter a great deal of data, but not all of it is necessary unless you are utilizing a professional grade CNC milling machine. For a hobby machine, many of the features that would use the data are not available anyway.

To create a new tool, in the Manufacture Workspace choose Manage→Tool Library. If you have multiple files open, select the one in which you want to save the custom tool data and then click the add button (plus symbol.) A pallet of different types of tools will appear and you can choose the one that matches your end mill (ball end mill, tapered, flat, etc.), you can also change the tool type later if you need to.

It's a good idea to start by entering text in the description field. Fusion360 will make some of the parameters you enter visible in the Tool Library to help identify a specific custom tool, but providing a few descriptive words can help you locate the tool faster.

The most important data to enter is the cutter data. A flute is a spiral cutting edge on the end mill. End mills come with different numbers of flues, common numbers are 2, 3, or 4. Geometry parameters include diameter of the cutting portion of the end mill (*i.e.* the portion that has flutes on it), the diameter of the shaft (which can sometimes be different from the diameter of the cutting portion), the overall length of the entire end mill, the length of the end mill that will extend beyond the holder, the flute length (*i.e.,* the length of the end mill that is covered by the flutes), and the shoulder length which is the flute length plus the length that transitions from having flutes to being a smooth shaft. Depending on the type of end mil, additional parameters may be needed such as the taper angle on a tapered end mill. When you click on each parameter, Fusion360 provides graphical and text help to make it easy to understand what each parameter actually indicates. With a pair of calipers it is relatively easy to enter all of the geometry data.

You can also enter data describing the holder. The holder, as the name implies, holds onto the end mill. Professional milling machines can be equipped with a carousel of tool holders, each holding a different end mill; in this case it is important to accurately enter tool holder information, but it is not essential for a hobby milling machine.

Speed and feed rates are important. It is possible to override them later when you are creating simulated tool paths so it's not necessary to enter that data now, but it's usually worth taking the time up front rather than having to re-enter the data for each new tool path later on. These data will depend on the material being milled, but for a hobby milling machine, that material will usually be aluminum. If you work with multiple materials you would create a different custom tools for the same physical end mill to correspond to each type of material.

Spindle speed indicates how quickly the end mill spins and is measured in units of frequency such as RPM. Surface speed indicates the speed at the outer edge of a flute, it is simply the circumference multiplied by the spindle speed and it is measured in units of inch/minute or similar. If you enter spindle speed, surface speed will be calculated automatically and vice versa. If the end mill were to move while spinning the actual speed along the surface would change (from a physics perspective), but that's not considered when defining surface speed as a parameter; we assume that the end mill is spinning in place (or that the outside edge is moving relative to the center point of the end mill).

Feed rate is the speed at which the end mill itself will move. As a parameter, this movement is irrespective of the spindle speed; in other words you can specify spindle speed and feed rate independently of one another.

The end mill can be moved in the x, y, and z direction and there are several terms that describe the type of motion. A plunge or plunging indicates a situation where the end mill cuts straight down into the aluminum in a manner similar to drilling a hole with a drill bit. Ramp or ramping is a combination of vertical and side-to-side motion. Lead-in and lead-out refers to motion of the end mill when it is not in contact with the aluminum (and will often be set to move faster as a result.) Feed rate by itself indicates motion parallel to the surface of the aluminum block, (*i.e.* motion on the x-y plane.) In general, ramping and plunging feed rates should be about one third the normal feed rate.

Fusion360 also has an option for a ramp spindle speed but most hobby milling machines require you to control the spindle speed by means of physically moving a belt between different pulleys, so the regular spindle speed and ramp spindle speed need to be the same.

Chip load is a measure of how thick the chips cut away from the block of aluminum will be. Chip load is not a parameter in Fusion 360 but it gives a relationship between the spindle speed and the feed rate: chip load = feed rate / (spindle speed * number of flutes) or feed rate = spindle seep * chip load * number of flutes. Generally, chip load will be decided upon based off the diameter of the end mill and type of material, spindle speed will be decided upon based on type of material and available spindle speeds for the milling machine (*i.e.,* belt and pulley combinations between motor and spindle), and finally the feed rate is calculated from the equation above. Much information is freely available on the internet to provide starting recommendations for chip load and spindle speed, but in practice choosing spindle speed and feed rate is more art than science due to the many unaccounted for factors that can affect the performance of the milling machine.

In a professional setting, optimizing the spindle speed and feed rate is important because if the feed rate is too fast the end mill could be damage, but if it is too slow unnecessary wear can occur as the end mill cuts many thin chips instead of fewer thick ones. For hobbyist milling machines that tend to use small diameter end mills, the end mills will most likely get broken long before it wears out so it’s a safer bet to go with a slower feed rate since this reduces the chance of breaking the end mill. Table 1 in the accompanying journal article gives values for spindle speed and feed rates used successfully by the author on a Taig model 2000 HD-ER micro mill, and may serve as a good starting point for experimentation with similar hobby milling machines. You may consider doing some test milling to find the values that work best for you before trying to mill out an injection mold.

2.2. Define the origin point relative to a simulated block of aluminum

The toolpaths describe the motion of the tool and therefore require a point of reference. This reference point will be the origin of the coordinate system used in the simulation as well as the origin for the coordinate system used in the real-world CNC machine. When you define the origin for the tool path simulation, you are defining the point on the real-world block of aluminum where you will zero the x, y, and z motion controllers.

In Fusion360, defining the origin is done in the Manufacture Workspace by choosing Setup→New Setup. A setup defines the size of the block of aluminum and the point on the block of aluminum that will represent the origin. You must create a setup before you can create a simulated toolpath, each toolpath is linked to a particular setup. You can have more than one setup associated with your Fusion360 project and a different setup could represent machining a new object on a second block of aluminum, or it could represent working on a different face of the same block of aluminum.

When you mill out your injection mold, you will start with aluminum bar stock of the correct width and thickness for your mold half. You will then need to cut the bar stock to length. For example if your mold half needs to be milled out of a 2 inch (50.8 mm) x 3 inch (76.2 mm) x 0.5 inch (12.7mm) block of aluminum you would use 2 inch (50.8 mm) x 0.5 inch (12.7mm) bar stock and cut it into 3 inch (76.2 mm) lengths. Ideally each piece would be exactly 3 inches (76.2 mm) long, but it may be that there is some variability in length. This variability could mean that the two mold halves don't line up perfectly when assembled. If pins are used to align the mold this is not a critical problem, but it is better if the two mold halves align up on at least one side of the mold. By choosing mirror image locations for the origin on each mold half, this alignment can be achieved, see Figure 17.

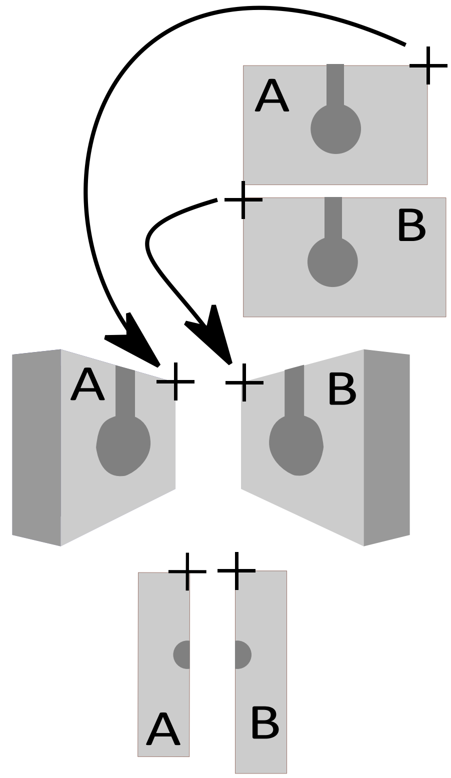

Figure 17. Crosses represent the origin point on mold half A and on mold half B. At the top of the figure the origin points are at the upper right and upper left on A and B respectively. The middle of the figure shows how the two mold halves line up. At the bottom of the figure, the top of the two mold halves are shown. Even though they are (unintentionally) of different lengths they still align at one side.

2.3. Simulate toolpaths for milling the cavities out of the block of aluminum with your end mills

Fusion 360 provides a number of different toolpaths to choose from. Each toolpath applies a different algorithm to the CAD model in order to create the specific path that the end mill will be moved through while cutting the aluminum block. The number of toolpaths to choose from and the number of parameters that control the algorithm for each toolpath can be overwhelming. Fortunately, there are many help features built into the user interface to aid you. Even with this help, in most cases selection of the best toolpath and parameters will involve a lot of trial and error.

A fairly common experience, especially when trying a different toolpath on a CAD model, is when the toolpath algorithm does not produce a toolpath at all. It occurs when the shape of the CAD model, selected end mill, and algorithm parameters result in no valid toolpaths. In some cases this outcome requires lengthy processing by the computer, which makes the situation all the more frustrating. There are too many parameters to discuss all of them here but Table 1 outlines some common problems that result in no toolpath being generated or an undesirable toolpath.

Table 1. Parameters frequently associated with a poor toolpath or no toolpath

Parameter	**Notes**
Tool selection	When machining pockets, internal islands, or grooves, if the end mill is too wide a toolpath cannot be generated. If you don't see a toolpath, choosing a very small diameter tool may help you troubleshoot the problem.
Machining Boundary	To run a toolpath algorithm, the face of the object to be machined must be chosen in the setup (see Section 2.2), however, it is possible to limit the machining operation to certain regions by setting the Machining Boundary. Setting the Machining Boundary is often necessary because different regions on your CAD model may be better suited for different toolpath algorithms. The boundary can be set by choosing a contour line on the CAD model. In some cases an adequate contour line cannot be found. In these cases, it is possible to select a line on a 2D sketch to serve as the boundary. To set the boundary you must use the Select option under Machining Boundary.
Tool Containment	Tool containment describes how the Machining Boundary should be interpreted. The tool can stay completely inside the boundary, the boundary can be for the center point of the tool, or the tool can stay completely outside. Cutting will still be based on the CAD model geometry, but in some cases the choice could cause the algorithm to fail to generate a toolpath.
Stepdown	Some toolpath algorithms cut into the block of metal through a series of progressively deeper passes, such as shown in Figure 5. Stepdown parameters relate to how far the end mill will be lowered into the sample after each pass. Some algorithms may have a Maximum Stepdown parameter and others may have a Minimum Stepdown. If not optimized the stepdown parameter could result in a curved surface that is too rough or too many stepdowns causing a longer machining time.
Stepover	Stepover determines how much overlap there is between parallel passes of the end mill. If the stepover parameter is too small many unnecessary passes could be made but if it is too large, there is a risk that the end mill could break while trying to cut too deeply into the aluminum block. In some cases a large stepover may also cause material that was supposed to be cut to be left behind.
Stock-to-leave	Some toolpath algorithms have the stock-to-leave parameter. The parameter is useful when you want to do a quick rough cut of the aluminum block followed by a slower more detailed cut to remove the final layer of aluminum. If the stock-to-leave parameter is not set to 0, in some cases it could result in no toolpath being generated. When sock-to-leave is the culprit, it often goes overlooked in the troubleshooting process causing much frustration. Checking that the stock to leave parameter is correct for each toolpath is a good habit.

Fusion360 has two general categories of toolpaths, 2D and 3D. The 2D algorithms do result in 3-dimensional machined objects, but represent milling operations where a 2-dimensional pattern is milled straight down into the aluminum block. The 3D algorithms are used when slopes and curved surfaces need to be cut into the aluminum block. Because injections molds usually consist of slopes and curved surfaces, 3D algorithms will be the most common type used.

After a toolpath is created it should show up in the screen as a line or series of connected lines showing where the end mill will be moved. A better way to visualize the toolpath is to simulate the cutting process and Fusion360 has a feature that allows this. With the toolpath selected in the Fusion360 Browser tree, select Actions->Simulate. The author finds it most useful to turn off the display of the tool and toolpath and to turn on the display for the stock material. When the simulation is running, you can see material being cut away from the simulated aluminum block.

2.4. Post-process the toolpaths into G-code that a CNC milling machine can read

A G-code file is an ASCII text file that contains commands that a CNC milling machine can read. Each command is represented by a letter, usually G, followed by a one or two digit integer. In some cases the G-code command is also fallowed by parameters, for example G01 X1.6021 Y-1.1496, is the G-code for displacing the end mill from its current position to 1.6021 inch (40.693 mm) in the positive x-direction and 1.1496 inch (29.200 mm) in the negative y-direction.

Fusion360 uses a piece of software called a “post processor” to convert a toolpath into the G-code equivalent that the CNC machine can read. Most of the G-code commands generated for a given toolpath will be identical from one CNC machine to another, but there will be some differences. Each CNC machine has different features and default settings that must be considered in order to create a working G-code file. Fusion360’s post processor allows you to choose from many different brands of CNC machines, but there may still be a need to manually edit the G-code to make it work with your CNC machine.

For the Taig model 2000 HD-ER micro mill, the “CNC Router Parts (Mach3Mill)/cncrouterparts” configuration was used. Figure 18 shows how the post processor output was modified by hand to be compatible. Most of the removed code was simply not needed, but the G28 G91 Z0 command resulted in the milling machine crashing the end mill into the part. For this reason it can be useful to zero the end mill well above the part for a practice run when you try out new G-code; any anomalies can be observed while the end mill is far enough away that collisions won’t occur. It’s worth pointing out that the simulation within Fusion360 cannot be used to identify these problems because they result from improper communication of the G-code command with the milling machine, not a problem in the original toolpath.

```
(2003 PRONGS 3 DEGREE BALL END MILL 6500RPM 2 AND .1 IPM UPPER RIGHT)
(T20  D=0.0279 CR=0.0148 TAPER=3.2DEG - ZMIN=-0.1969 - TAPERED MILL)
G90 G94 G91.1 G40 G49 G17
G20
G28 G91 Z0.
G90

(POCKET-PRONGS)
M5
M9
T20 M6
(TAPERED BALL END MILL .75MM  6500RPM 2 AND 0.1 IPM)
S6500 M3
G54
M9
G0 X-1.656 Y-0.7901
G43 Z0.6 H0
Z0.0794
G1 Z0.0448 F0.1
X-1.6559 Z0.0442
X-1.6558 Y-0.79 Z0.0436
X-1.6555 Y-0.7899 Z0.0431
G3 X-1.6551 Y-0.7897 Z0.0426 I-0.0017 J0.0031
X-1.6547 Y-0.7893 Z0.0423 I-0.0021 J0.0028
X-1.6543 Y-0.7889 Z0.0421 I-0.0024 J0.0025
```

Replace grayed section with:

```
G0
Z0.0794
X-1.656
Y-0.7901
```

```
G1 X-1.5094 Y-0.8444 Z-0.1953
X-1.5093 Y-0.8446 Z-0.1947
X-1.5092 Z-0.1941
G0 Z0.6

M9
G28 G91 Z0.
G28 X0. Y0.
M30
```

```
Delete this grayed section.
```

Figure 18. Illustration of how the G-code generated by the Fusion360 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 CNC 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 ½ inch (12.7 mm) block should be used. The author used a chop saw to cut aluminum bar stock to the appropriate size. To make smoother edges, the aluminum was cut slightly longer than needed and then faced off using the milling machine.

The Taig CNC milling machine comes with a vice to clamp the aluminum block into place. Clamping is straight forward, but there are some considerations that must be discussed. First, it's important to position the block so that the end mill will not come into contact with the vice while milling. For creating an injection mold, the end mill will primarily be cutting out pockets and drilling holes so it will not approach the edge of the aluminum block where it might contact the vice. However, there are still some cases where it can contact the vice, for example when cutting the port on the side of the mold half, or when facing off the rough cut edges of the aluminum block (as mentioned above.)

The second consideration is to make certain that the aluminum block is secure. If the block comes loose during milling it may ride up on the end mill, potentially breaking the end mill. Even if the end mill is spared it will at the very least make a bad cut and probably ruin the part.

The final consideration is that the surface of the aluminum block must be perpendicular to the end mill in order for the toolpath captured in G-code to accurately cut. If the surface is tilted slightly, the CNC milling machine will not detect this and will simply make the cuts assuming it is perpendicular. To avoid this problem, a right angle gauge can be used to check the surface against the end mill before cutting.

3.2. Insert the end mill into the spindle.

Professional CNC milling machines have automated tool changers and can hold multiple end mills within the machine such that G-code calls can be used to select the appropriate end mill for the cutting job. Hobby milling machines don't have this option, so each time you need a different end mill you must remove the previous one and insert a new one. The Taig CNC milling machine uses the common system of securing the end mill with a collet. The collet is a collar-like metal piece that the end mill is inserted into. The collet and end mill are then inserted into the spindle and a threaded cap is tightened to create a clamping force between the collet, the spindle and the end mill. The collet must be chosen so that it matches the diameter of the end mill shaft.

The Taig milling machine uses a pair of stepped pulleys connected by a belt to determine the RPM of the spindle. Depending on the desired speed for the end mill, it may be necessary to move the belt to a different pair of pulley steps when the end mill is changed out.

For safety, the milling machine should be turned off and unplugged from power before changing the end mill and adjusting the pulleys.

3.3. Move the tip of your end mill to the origin point you defined in the CAD software and zero the CNC mill

Using the Mach III software, the end mill must be moved to the origin point specified in the setup in Fusion 360. The position of the end mill can be controlled via the arrow keys on the keyboard. To control how quickly the end mill moves, the jog rate can be increased or decreased. To precisely position the end mill, the jog rate should be set very low. The center of the tip of the end mill needs to touch the origin point. Assuming the origin point is the upper left corner of the aluminum block, the following procedure can be used:

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 finder, a tool that is inserted into the collet and spindle instead of the end mill. The use of a center finder required more steps and ultimately wasn't necessary for making injection molds.

3.4. Run the G-code software to begin cutting the aluminum

Once the end mill is correctly centered, the G-code file can be loaded and run using the Mach III software. As was mentioned previously, it's a good idea to zero the z-axis above the workpiece and give the G-code a dry run before trying to cut your aluminum block. Mach III also has the ability to simulate the G-code prior to running. The simulation may allow you to catch potential problems and it will estimate the time required to complete the job on the CNC milling machine. When milling curved surfaces with small stepdowns, which injection molds often have, the machine may need to make many passes which can result in run times lasting multiple hours. In these cases it is useful to know the estimated run time in advance.

3.5. Add lubricant to prevent overheating and vacuum away chips as the mill works

To prevent the end mill from overheating and wearing out, lubricant is added while it is cutting the aluminum block. Professional machines have built in systems to spray lubricant on the part during machining and the lubricant is captured, filtered, and reused. Hobby milling machines do not have this feature, but periodically spraying WD-40 onto the end mill and aluminum block provides adequate lubrication.

The milling process generates a large amount of aluminum chips. A shop vac with crevice tool attachment was used to remove the chips. The crevice tool was held close to the end mill to suck off chips while milling. The author found that it was generally unnecessary to apply constant vacuum; instead the vacuum was used occasionally during a run to clear off chips so that lubricant could be sprayed on the end mill and aluminum surface without interference from the chips. Due to the amount of chips generated, it is important to vacuum frequently to keep the work area tidy.

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 cleaned 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

Alignment pins were cut to length with a hack saw from aluminum rod. Rough edges were removed with sand paper. In most cases the author did not affix the alignment pins to the mold, but this sometimes resulted in pins being dropped to the floor and lost. Better results can be had by using high temperature adhesive to affix the pins into the alignment holes on one mold half which will prevent them from falling out when the mold is separated.

FINAL COMMENTS

This guide is intended to be a starting point for readers interested in creating aluminum injection molds, but there is room for improvement and the author hopes that others will be inspired to expand and improve on the ideas and techniques shared here.