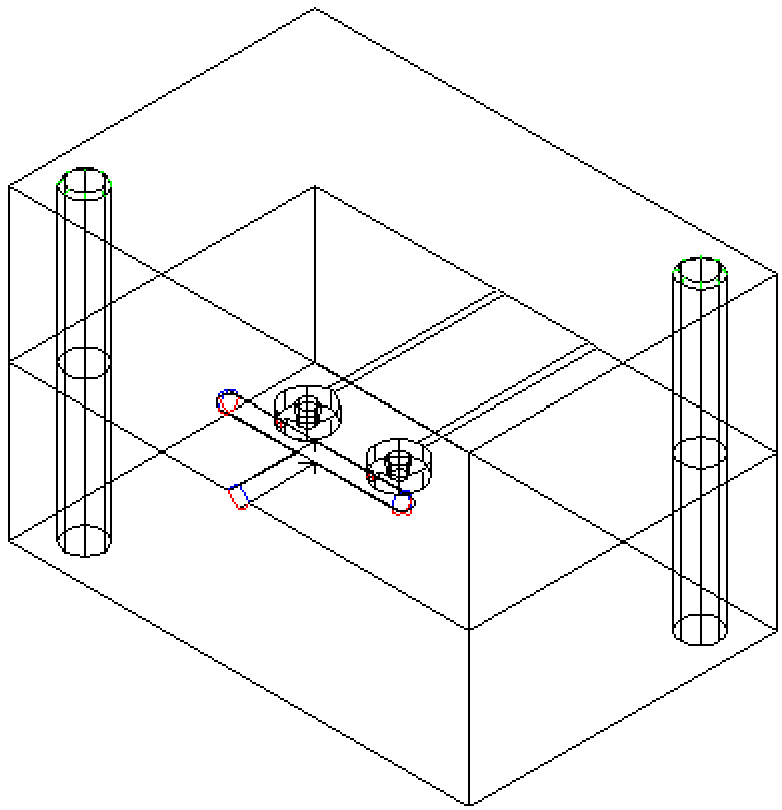
Of Runners, Gates and Cold well plugs:

Kenneth Y. Maxon



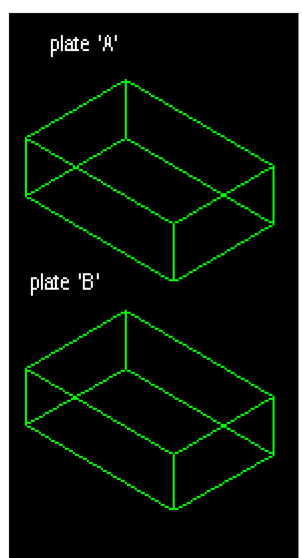
An over view article on minimal injection molding

So you need to make a distinctively large quantity of plastic parts in your next robotics project. Should you simplify the design to save your self the trouble? There are several options available to you. Many of these fall into the categories of hand forming (heat & bend, file drill.), vacuum forming, conventional milling, CNC milling, injection molding, and sls stereo lithography categories. *(lets face it, if you can afford stereo lithography, you can afford to just pay somebody else to make the parts for you.)* The order these options is presented is not coincidental. Generally this is directly tied to the dollar value associated with each of the processies. But when and where are these monetary lines drawn? There are several factors to consider. Of coarse, money is a driving force, but how often do we undervalue our own time spent in manufacturing a piece for our robots. Sure it cost $5.00 in raw materials, but I then invested seventy hours over three weeks to manufacture the end product. Another consideration is that of repeatability. Will one piece be exactly the same as the next, or will I have to make special considerations for each piece in my design. The proceeding ideas represent decisions that we as robotics enthusiasts / hobbyists make normally in a subconscious manner. I present the following based on my recent decision to choose one of the fabrication techniques (Injection molding), which, aided by modern technology proved to be rather painless. Specifically, this tends to be the proverbial `path less chosen' that I would urge more people to strongly investigate.

In choosing to injection mold the tractor tread pieces for my robot I was taking a large step towards reducing my overall project cost. Specifically, when accounting for my time in machining complex 3-D pieces, even worse, one hundred and forty four of them, one begins to look for other options. The argument proceeds that this is why we have CNC machining. The problem becomes more evident when you take into account the actual part geometry. The piece I dreamed of would require four separate setups in CNC machining and as further investigation proved require producing four separate sets of custom machined complex tooling fixtures to hold onto the part while machining it. So I decided that a injection mold would be the preferred method of part production.

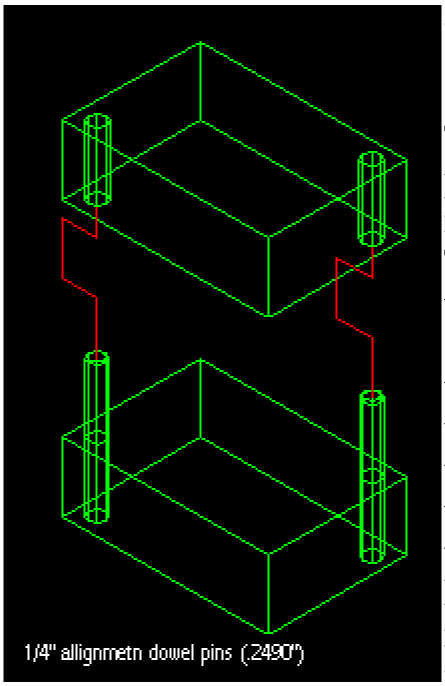
Upon choosing this method of production, advice will flow in. `That technology is for large companies making tens of thousands of parts', they'll say. Not to discount this wonderful advice, in large part this is true, but not in the sense that it applies to the common robotics hobbyist. There are many small shops that are more than willing to do small production runs in molding that are willing and friendly enough to stop and help the little guys. Right here in West Seattle, I have had the good fortune to work with a small company called N.W. Shortline. Named after the old rail road line running through the area. As a matter of fact they make small production runs of specialized model train parts sold exclusively through external vendors and mail order. In other words they make a business of molding small plastic parts when only a hundred or so are needed. Exactly what we're looking for.

Of molds, sprews, locating rings, return pins, ejector pins, cold well plugs, waterlines, pipe threads and a whole specialized industry which revolves around them. The terminology is overwhelming, but most of this applies to a full sized `production' mold. We want simple, cheap and cost effective. The remainder of this article is dedicated to the **minimums** that one requires to get a small part manufactured.



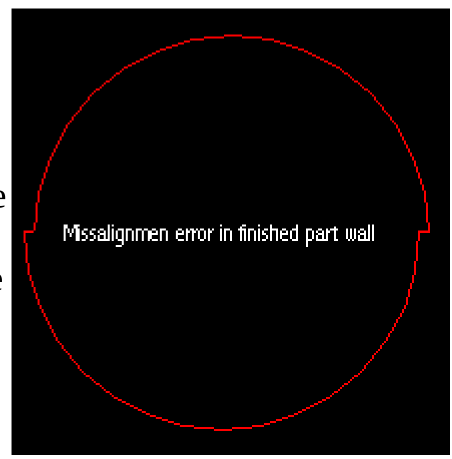
Plates:

Simple molds can be made from as little as four pieces of material, yes I said four. Before one procedes to far make sure to contact your molder and find out the exact dimensions of the mold you need to make. (many of the different injection mold machines require different sized molds.) Specifically I found that I needed to produce a mold with outside dimensions of 2"x2"x3". Right away, this defined the stock size for the geometry of two of the above mentioned four Pieces. These pieces form the *A* and *B plates* of the cavity. (fig. 1) In small production, the material of choice for mold production is generally aluminum 6061-T6 (<1000pcs) or 7075-T6 (1000-5000pcs). Additionally surface treatment such as anodization provided by several local shops aprox $70 although with these small lots and not caring what color you get they'll often throw your job in with another corporate paying company and not charge a penny. Note that anodizing your mold is not required and in most `small' jobs (<2000pcs) is simply a waste of money. We'll come back to the A and B plates later, but for now let's discuss the other two pieces needed for a successful minimal injection mold.



Pins:

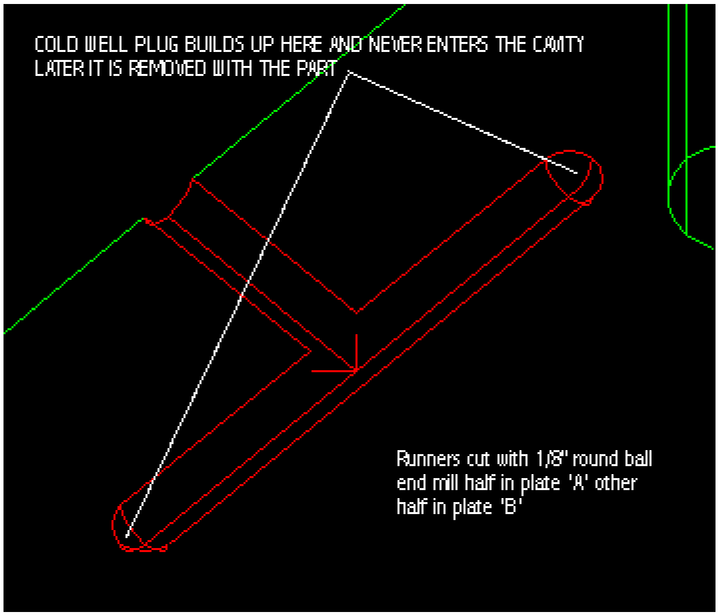
The two additional pieces required are the alignment pins. Alignment pins protrude from the A plate of the mold and extend into a matching orifice on the B plate. The pins ensure the geometry lines up properly when the plates are pressed together. (fig. 2, left) In some small molds these are cut directly into the geometry of the A or B plate. If you choose to produce the pins in this method, make sure to buy aluminum thick enough to encompass them. I opted however to use *dowel pins* as this afforded me theoption to press them back out at a later date and replace them. (fig. 3, right shows common miss-alignment problem) Additionally, dowel pins are made from an incredibly hard steel which does not wear, and more importantly one can cheaply purchase a reamer($8- $15) to make the receiving hole an exacting size for the dowel pin. A method which involves three steps, center drill to locate the hole, drill through (undersize), and reaming the hole to size produces a precision located hole of propper size while minimizing on slop in mold alignment.



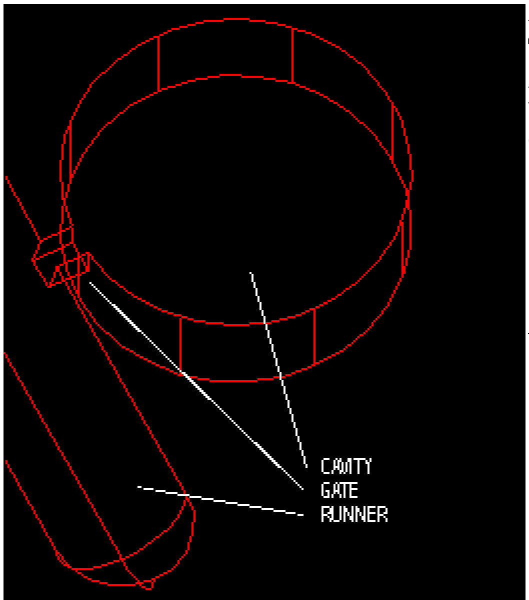
More on Plates:

Now we have defined the four pieces that make up a **minimal** injection mold, one is left wondering, wait there must be more? There is more, much more, but in the minimal mold, not too much. All of the additional work in our minimal mold will be involved with shaping of the A and B plates we discussed previously. The face where the A & B plates meet is referred to as the parting plane of the mold. Care must be taken to make sure that these two surfaces are machined flat. In high end molding, the two pieces are actually burned together through EDM machining processes as a last step to insure a good fit. (other options such as surface grinding are used as well.) For the prototype it is enough to machine them flat and let the people shooting your mold know so that they don't over pressure the mold.

Runners:



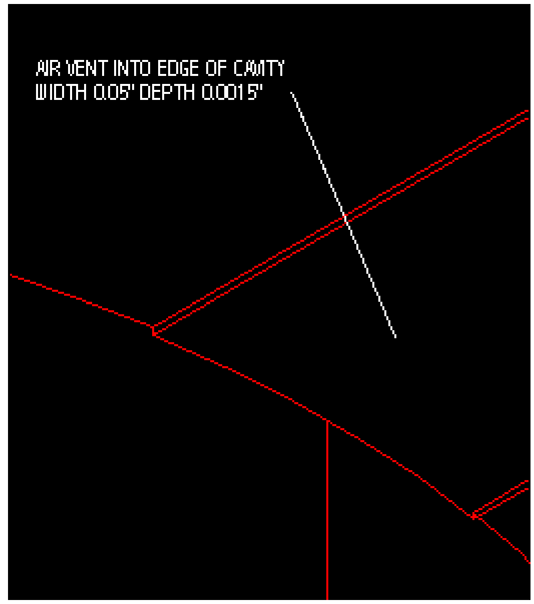
Into both the A and B faces of the mold a semicircular channel called the *runner* is cut. (fig. 5) This channel anywhere from 1/8" to 1/4" becomes the path that the plastic along which the plastic flows when entering the mold from the machine. In roughly the middle of the mold we will cut a `pocket', in the shape of the part we wish to produce. This pocket, the *cavity* as it's called provides a spot for the plastic to cool into our part. So the runners carry the melted plastic from the machine to the pocket, but they don't just dump it into the cavity, rather they stop just past it. Yes, that is right, the runners carry the molten plastic right past our cavity. The runner dead ends in and area called a *cold well*. As molten plastic travels into our mold and down the runners under substantial pressure, area in contact with the walls of the runner begin to cool slightly and the leading plastic builds up a plug of cooled plastic. The plug more properly called a *cold plug* flows right past the small entrance to our mold cavity, and gets trapped in the cold well.



Gates:

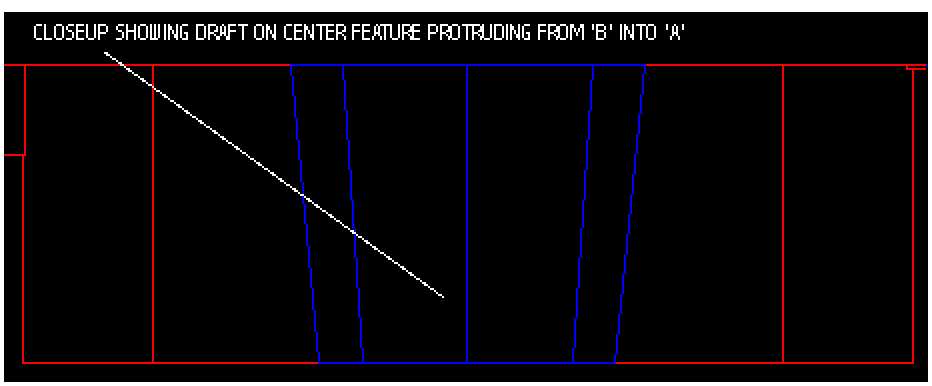
A very small notch is cut between the runner channel and the cavity itself. The runners are cut into both the A and B plates, but the small notch called the *gate*, is cut into only one. This gate breaks into the mold about one quarter of an inch before the end of the runner, thus forming the cold well mentioned in the preceding section. Specifically, the gate should measure .03"x.03" - .04"x.04". (fig. 6) This causes the plastic to be squirted into the cavity, and causes a bit of turbulence in the molten plastic to ensure full remix upon cavity purchase.

Air Vents:

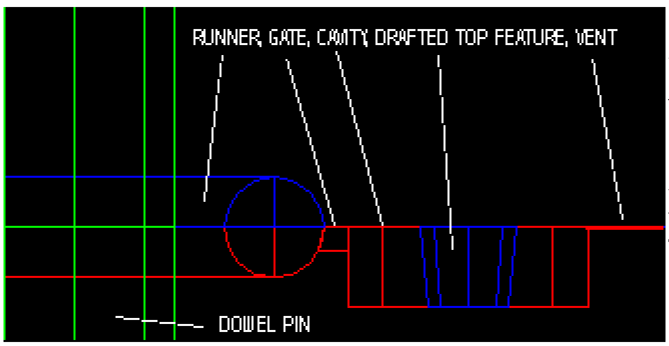


When the plastic is injected into the mold cavity, pressure builds up to a point high enough that it can rupture thin walls in the mold. The air in the cavity at the start of the cycle must be allowed to escape through *air vents* small enough that the plastic will not follow. Typically, between .001" & .002" deep, small slots are cut into the face on one side of the mold (A or B your choice) to allow air to escape. (fig. 7, the depicted slot is .0015" deep and .05" wide leaving to the right of the diagram) This also helps keep air bubbles from forming in the final part. There are two factors that keep the plastic from flowing out of the air vents under considerable pressure. 1. The very small cross sectional area of the vent itself acting in conjunction with the surface tension of the plastic. 2. Vents are always located across the cavity from the gates, and as the molten plastic flows in along the cavity walls a thin skin forms which effectively blocks the air vents when the cavity is almost full! The skin effect of the plastic adds to the characteristic shinyness and outer layer hardness of an injection molded part.

Draft:

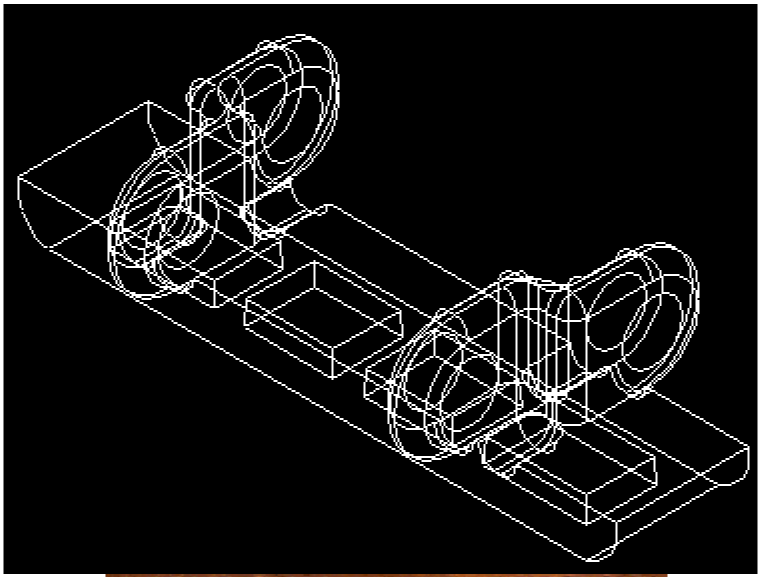


Finally, detailed attention must be paid to the walls of the cavity. The concept of *draft* is to machine all of the near vertical walls in the cavity with a bit of an angle. (fig. 10, note that the piece sticking down into the part to form the center hole {blue} of this 'washer' actually sticks out of the other half of the mold!) As plastics sublime, they tend to shrink, as much as .01" per inch. (.003" to .006" per inch is common however) This means that the part will tend to pull inwards, away from the outer edges of the cavity for easy extraction. Upon first review this sounds great, but the inward shrinking of the plastic causes it to grab onto any "islands" or protrusions sticking up into the cavity. (fig. 11) Thusly, drafted walls on near vertical cavity protrusions (islands) is a requirement, unless one molds extremely small parts. (less than .2" tall) There are two simple solutions to drafting vertical surfaces. 1.) Use a drafted cutter which is pre-ground with 1,2,3. degrees of draft. 2.) Use a CNC program to generate milling code with effectively drives down the angle of draft. Other more complex methods of tilting the head on manual machine or sitting down with an old fashioned file are not unheard of however.

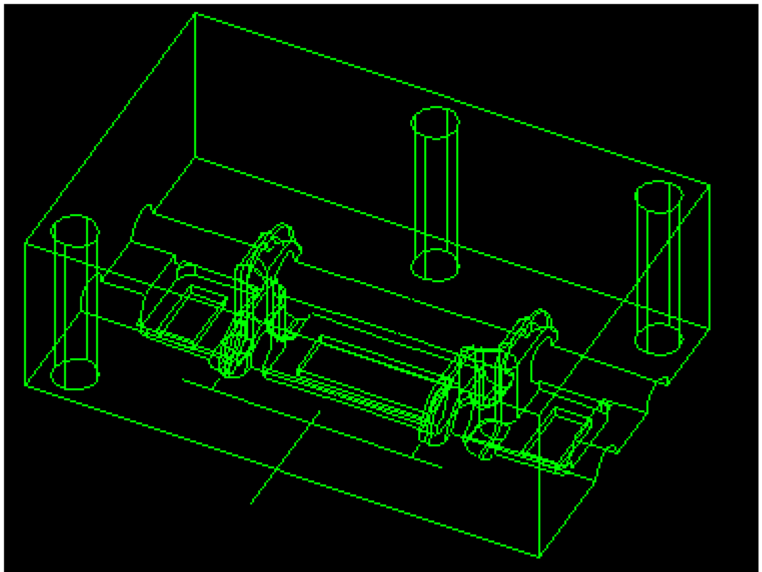


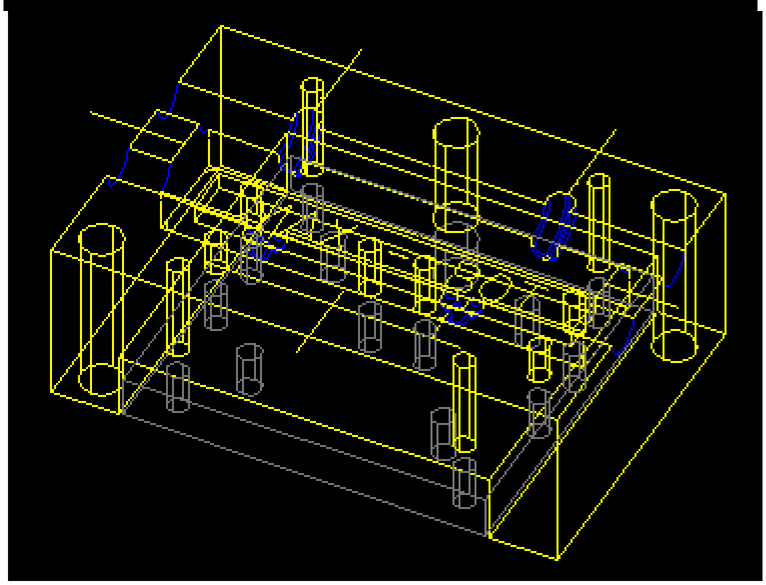
Putting it all together. Figure #9 [SIDE VIEW] shows the A & B plates of a small mold which might be used to produce a small washer. Note detail 1-4, the runner, gate, cavity w/draft, and air vents respectively. These are what I've termed simple or minimal injection molds. It is not uncommon to find 700 page volumes on the subject of injection molding that do not exhaustively cover the topic. This overview is however enough to introduce minimal mold construction for lots of the small parts commonly needed in hobby robotics.

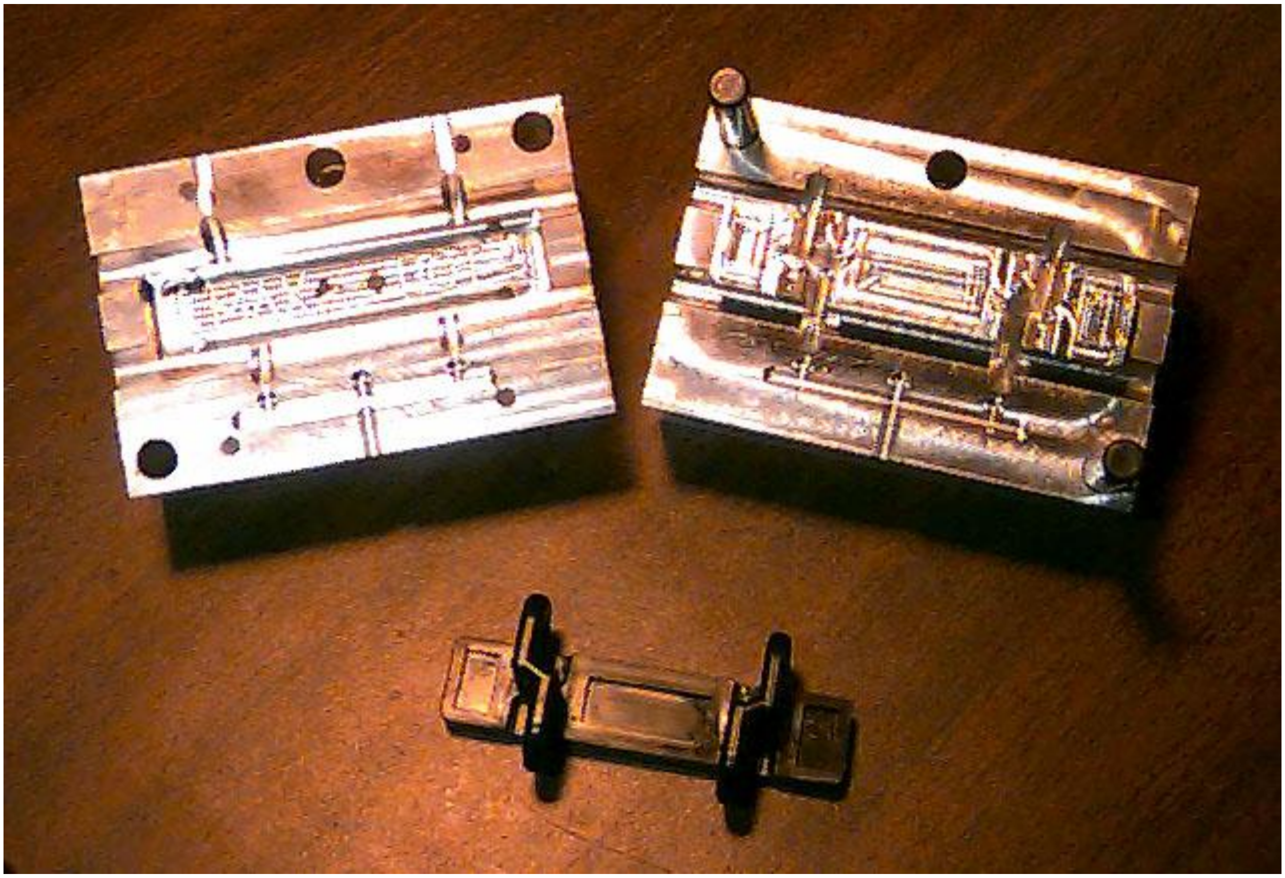
You don't need much imagination to look around your kitchen and find plastic injection molded parts that cannot be made in such simple manners but that will lead to the next subject of ejector plates and ejector pins, a topic that just enhances what we've already discussed so far. (another article if anybody wants it) Just remember, the processesies outlined above work for planar parting lines and molds that open simply. (I.E. ortho-normal to the parting plane) The final figures depicts the cad representation of the parts which I molded. These parts have non-planar parting geometry's, and removable plungers used to make undercut hollows (sideways through holes) in the parts. I've made the 3-D wire frame CAD geometry for this file available on my web site for downloading through `(http://www.oz.net/~kmax)` see below.

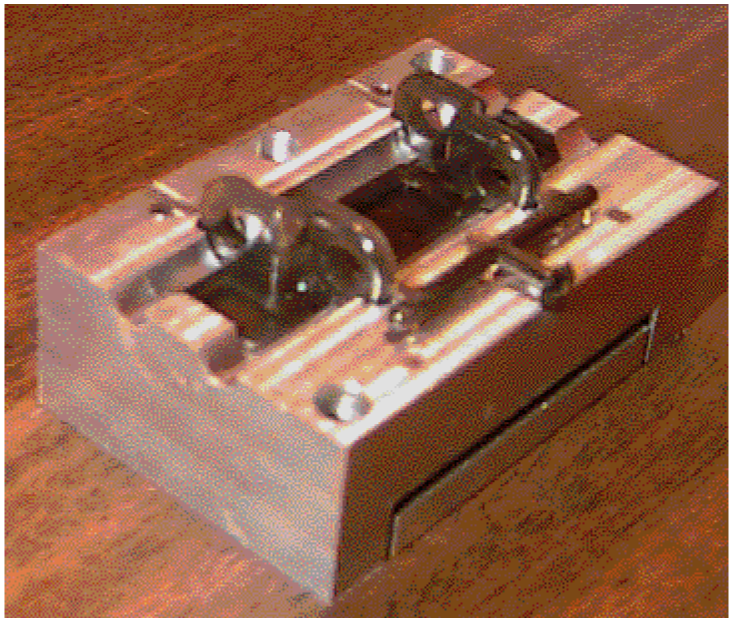


These pictures depict the tractor tread pieces I am making. See if you can locate the different parts discussed above (Runner, Gate, Cavity, ...)









Many of the SRS members have the abilities and machinery necessary to make minimal injection molds. The next time the need arises for numerous plastic parts weather spacers, washers, Lego blocks, or tractor treads I hope you will consider this as a viable option to choose from your toolbox.