(05 November 2008)

I am now in full production on the blocks. Finally! Although there are other parts being made in preparation for finished castings, it was absolutely imperative that any and all changes were documented and tested prior to the final machining process. Each block must have a starting reference point which all other measurements are taken from. This must be taken seriously and machined exactly to the prints. Keep in mind, that if this is not done correctly, then the main bearings that support the crankshaft may not be in the exact location. If it is higher or lower, then the compression ratio, deck height, head placement, intake manifold holes, etc. would all need to be modified. Not good! This is called the "trickle down effect". A good example is if the compression ratio is not correct, then do I change the height of the wrist pin in the rods, or raise or lower the wrist pin hole in the pistons, or make the top of the piston lower or higher. Enough of the "what if" scenario. Once the top of each block has been machined, the next operation is to machine the complete bottom of each block. Picture #1 shows the before and after blocks. while picture #2 shows all the blocks with the finish bottoms. I would also like to add that I am now doing what is called "thread forming". This is when a special tap is used to displace the metal and actually push the "valleys" into "peaks" which form the threads. This makes a thread about 20% stronger. The other added advantage is, there are no metal chips, because no metal has been removed! My CNC machining center will tap holes at 3,000 rpm. In less than one second a 5-40 can be tapped .375 deep. The machine can tap almost any size hole and there are a lot of 2-56 holes throughout the engine. Pic #4 shows a close-up of the threaded holes and the side machining of the main bearing supports. Since there are variables in the castings the center distance and width of the main bearing supports, are critical.

Now may be a good time to give a short explanation and an understanding of the programming information and complexity, which must be absolutely correct. If you look at picture #4 you will see a machined arc on the front surface of the block, directly below the four tapped holes. Look at picture #3 and you will see how the block is mounted in the jig. The CNC milling machine is capable of cutting an arc of about any diameter, by simultaneously using any of the two the three axes available. This particular move necessitated the movement in a Y - Z axes. Remember X is left and right, Y is away from you to towards you, and Z is up and down. This is only a partial list of things that need to be programmed before a single cut can be made: diameter of cutter, speed, feed rate, starting point, ending point, center of arc, direction of arc. and coolant on. A basic line of code to cut just one arc would look something like this:

G0 Z.1 M8 (rapid Z down to .1 above the part and turn on coolant)

G1 Z0 (this takes the Z to the beginning of the cut)

G2 Y-.325 Z0 I-.325 J0 (this move cuts and arc in the clockwise direction with a finish point of X.275 - Y-.325)

G0 Z.1(this rapids the cutter up to .10 above the starting point)

The width of the main bearings supports on the bottom of the block have 10 arcs, with half being clockwise and the other half counter-clockwise, which bring the width to exactly .244. There are also 5 arcs to remove excess metal from the center of each bearing support. Later on a .875 ball end mill is used to remove the remaining material from the middle of each main. If just one digit is off, or a decimal point put in the wrong place, the the entire casting can be destroyed, almost instantly. Remember the saying, "garbage in - garbage out", this is absolutely true when programming any piece of CNC machinery. If just one block is destroyed, the net loss is over $5,200.00. The actual piece obviously did not cost that much, but a complete engine is no

longer available. You can now see how important that everything be documented and tested prior to starting the machining operations.

I hope this is helpful in trying to explain why it sometimes takes a considerable amount of effort to proof a program. Once the program is running smoothly, then I go back to see if there are areas where some time can be eliminated, by cutter selection, speeds, or feeds. There are at least 42 holes in the bottom. Each hole must be center drilled, drilled, then tapped. If only 2 seconds can be removed per hole, this is a net savings of 84 seconds. If you multiply this times 40 blocks you get 3360 seconds, or 56 minutes. This relates to almost an hour saved on just one operation.

Pic #1

Pic #2

Pic #3

Pic #4