

Drawing a Clock wheel with Microstation – Straight Spokes

This is a beginner’s tutorial on drawing a clock wheel in CAD using Bentley Microstation. The DXF format of the drawing is used in a CNC milling system to cut out the sections to form spokes (a process called “crossing out” by clockmakers). There are several designs of wheel spoke: straight, tapered and, for lack of a better description, spokes bounded by hyperbola shapes. The example used here is the third wheel from John Wilding’s manual on How to Make an English Regulator Clock. This and other wheels in the clock have straight spokes.

The first task is to set up Microstation’s Tools and toolbars. Click the main menu’s Tools for a drop down menu. Check the first 5 and the View toolbar. See Figure 1

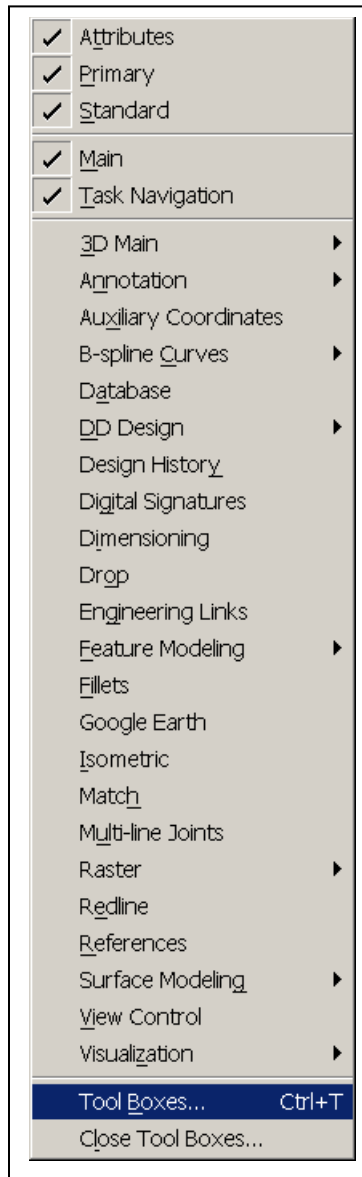


Fig. 1

The next step is to click Tool Boxes – the second to the bottom choice. This opens a Tool Boxes window where you check the toolbars you need. Most are unchecked. Check the following: Attributes, Element Selection, Ellipses, Fence, Linear Elements, Main, Main Classic, Manipulate, Measure, Modify, Primary Tools, Standard, Task Navigation and View Control. Check also the boxes for Show Tool tips and Immediately Open. See Figure 2

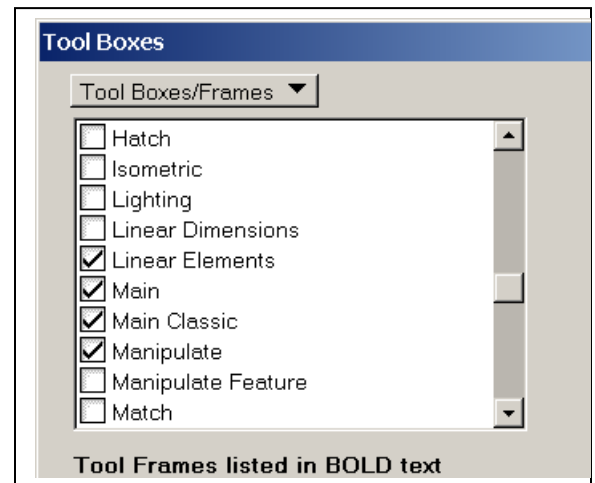


Fig. 2

The next step is to fill in the DGN File Settings. These settings determine the grid, which makes drawing precise dimensions much easier. Click Settings in the main menu, click on Grid about midway down and this brings up the Modify Grid Settings. The major grid setting is shown by lines, the minor grid settings are shown by dots. For drawing a clock wheel with various measurements of 1/8 inch or more, a setting of 16 is sufficient. For measurements in 16ths, a setting of 32 is appropriate. The Grid Master cell is filled out with the decimal fraction of 1/32 in this case. We will use 32 and .0313 although to make a spoke exactly 3/32 wide, we really need to use settings of 64 but the dots become difficult to see. We will split 1/32 divisions in half by eye to form a 3/32 spoke width.

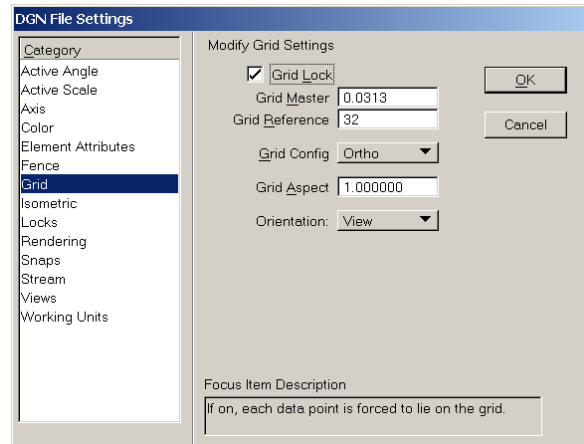


Fig. 3

For an example of the toolbars you will be using, see Figure 4.

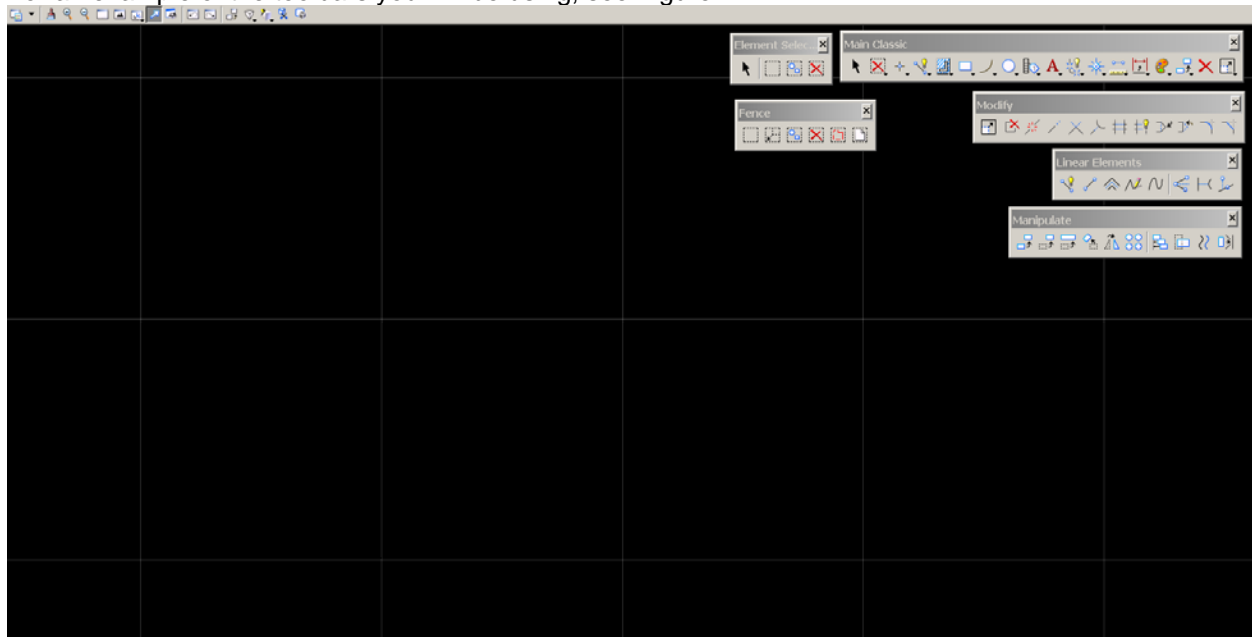
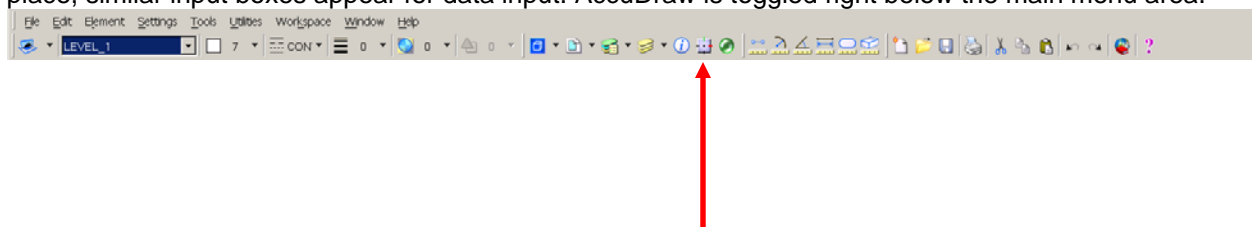


Fig. 4

Be sure to toggle the AccuDraw icon to turn AccuDraw on. For our purposes, this allows you to input exact diameters for your circles. The icon is a toggle. To determine if it is on, you should see three cells at the bottom of the screen showing the cursor location. With AccuDraw on, when you select an element to place, similar input boxes appear for data input. AccuDraw is toggled right below the main menu area.



If you are going to draw a number of wheels and other clock parts to be cut out via CNC, it would be wise to create a table or spreadsheet of measurements to prevent having to look up drawings in the actual manual again and again. See Figure 5 for an example.

5/23/2010

Regulator	Thick	O.D-In	Bore	Teeth	Rim	Boss	Spokes	DGN-File-Name	Gcode	Cut-out	Teeth Cut
Barrel-Ratchet-flange	1/4	2	5/16	50				ER-BarrelFlanges	Yes	Yes	
Barrel-Flange	3/16	2	5/16	NA				ER-BarrelFlanges	Yes		NA
Main-wheel	3/16	3-1/2	5/16	144	NA	NA	NA	ER-MainWheel	Yes		
Hour-wheel	1/16	3-1/2	5/16	144	1/4	1	1/8	ER-HourWheel-102509.DGN	Yes	Yes	
Center	1/16	2.35	3/16	96	3/16	9/16	3/32	ER-CenterWheel	Yes	?	
Third	1/16	2.21	3/16	90	5/32	9/16	3/32	ER-ThirdWheel	Yes	Yes	
Escape-Wheel-/spare	3/64	1-3/4	3/16	30	7/32	1/2	1/16	ER-EscapeWheel	Yes	Yes-2	
Maintaining-ratchet	3/32	3-1/4	5/16	120			NA	ER-MaintainingRatchet	Yes	Yes	
Weight-tray	1/16	2-1/2	2-parts	NA							
Slip-washer	1/16	1									
Pulley	1/8	2-1/4	7/32	NA	1/8	1/2	1/8	ER-WeightPulley	Yes	Yes	NA
Pulley-frames	1/8							ER-PulleyFrames	Yes?	Yes-2	NA
Weight-top-end-cap-Fit-in-tube	1/16	1/3/4	7/16	NA							
Weight-bottom-end-cap	1/16	1-3/16	#27	NA							
Maintaining-ratchet-detent--mild-steel	1/16							ER-MaintainingRatchetDetent	Yes		
Back-cock								ER-Backcock			
Top-of-Frames-design	1/8							ER-TopofPlates-cutout	Yes-2		

Section Break (Next Page)

Fig. 5

Beginning the Drawing

The plan is not to draw the entire wheel but draw only a section of it and rotate that to form the entire wheel. We begin by placing circles using 0,0 as the center of the wheel. We place circles for the I.D., boss O.D., the inside of the rim and the O.D. See Figure 6

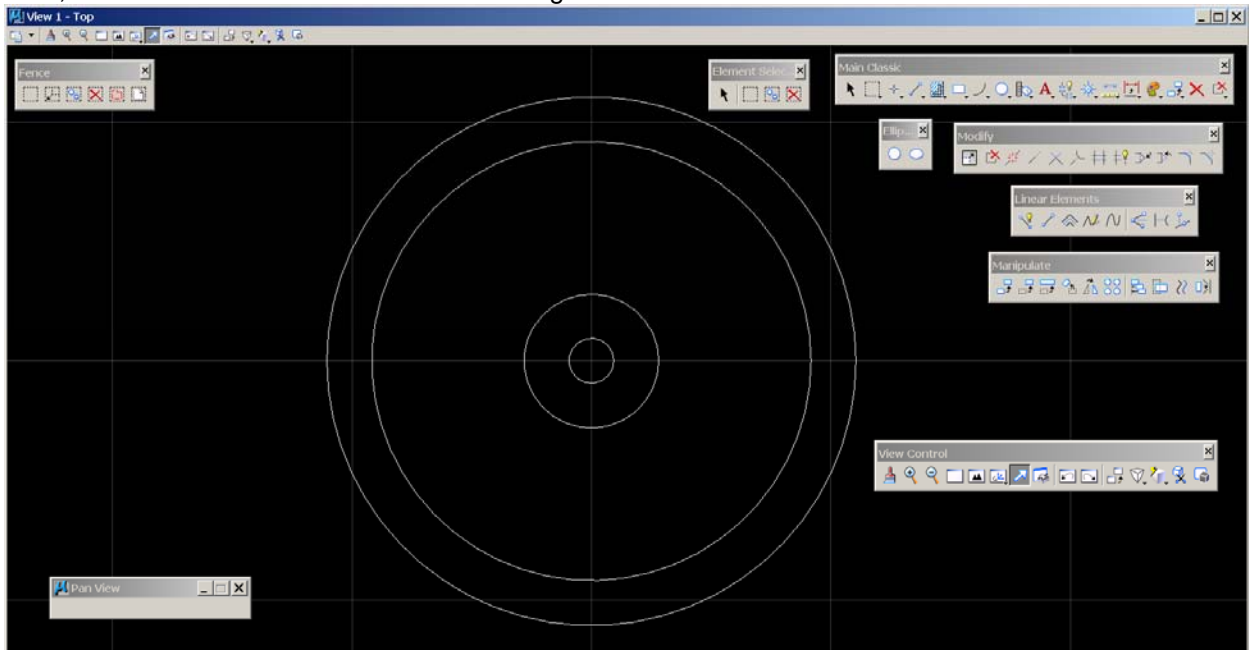


Fig. 6

We then place two lines forming one of 6 spokes, this one at the very top of the wheel. Here we have to estimate the space between two dots 1/32 apart on either side of the vertical line at X=0. See Figure 7.

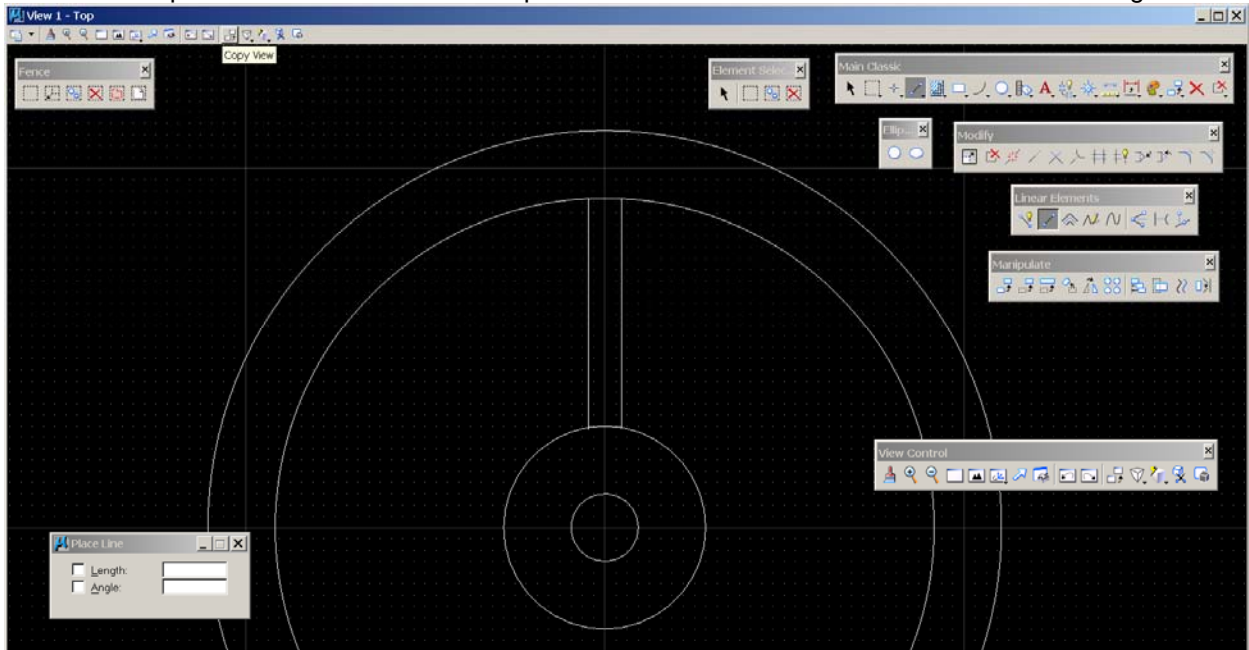


Fig. 7

Place the fence around just the two vertical lines and click the rotate icon in the Manipulate Toolbar. A window opens up. check Copies, Check Use Fence, enter 60 degrees, hit Enter and place the cursor in the center of the wheel at 0,0. You will see the two lines at the left of center in the fence. Left click to set the lines at the new location and right click to fix the position and end the rotation. See Figure 8 for a view of placing the fence and Figure 9 for the fence at the new position.

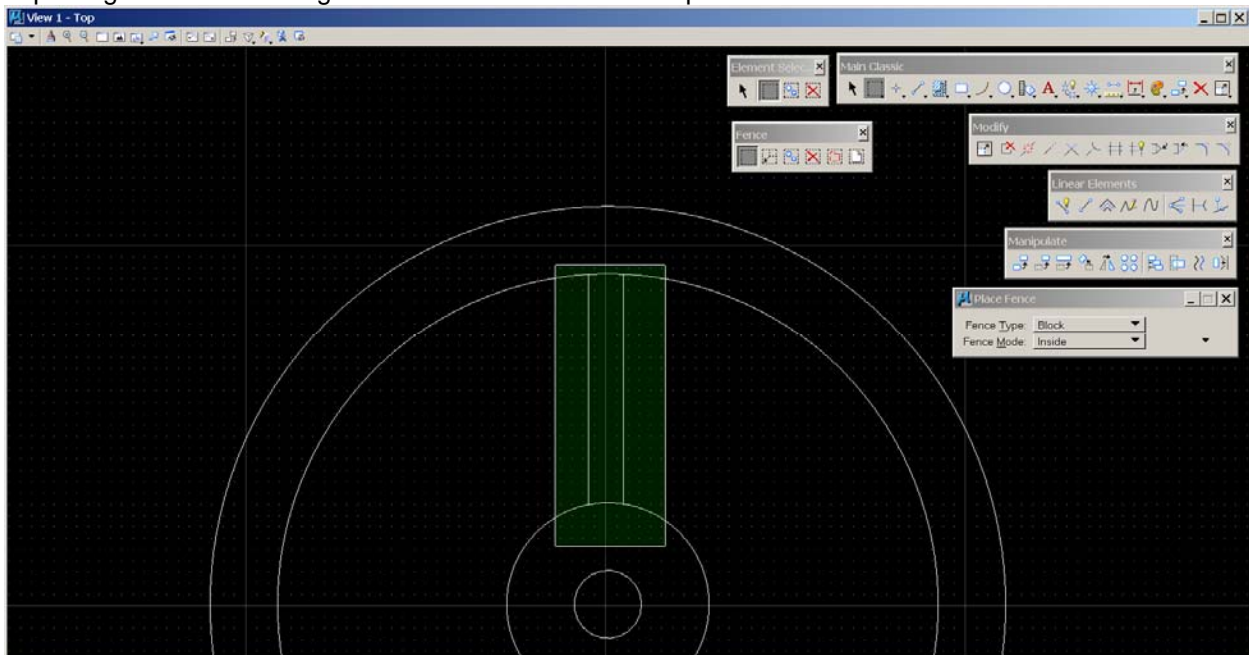


Fig. 8

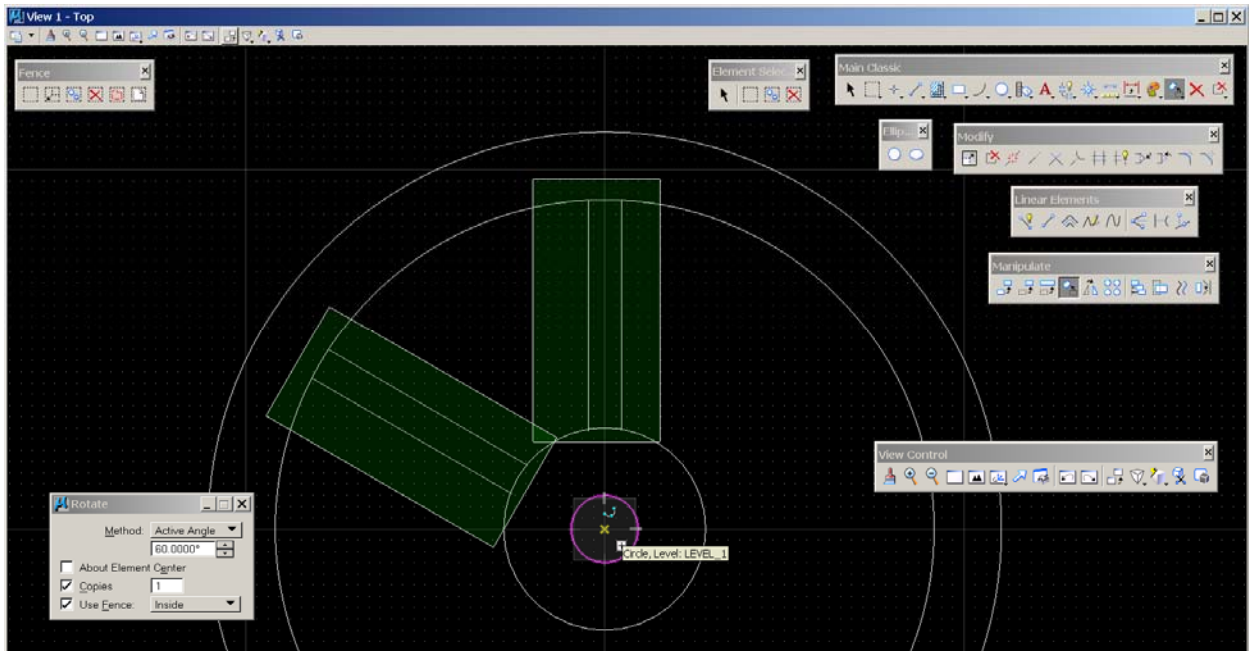


Fig. 9

Forming a Section of the Wheel

We next form one of 6 sections to be cut out of the brass blank to form the spokes. We need create only one section. Once completed, we simply rotate it as we did the two lines. To create one section, we use the Partial Delete icon in the Modify Toolbar – the second icon in from the left. We need to create a closed shape by partially deleting the lines where the spokes connect to the boss and to the rim. By clicking on a line and moving the cursor, we erase part of a line. See Figure 10 for a view of how little is involved here.

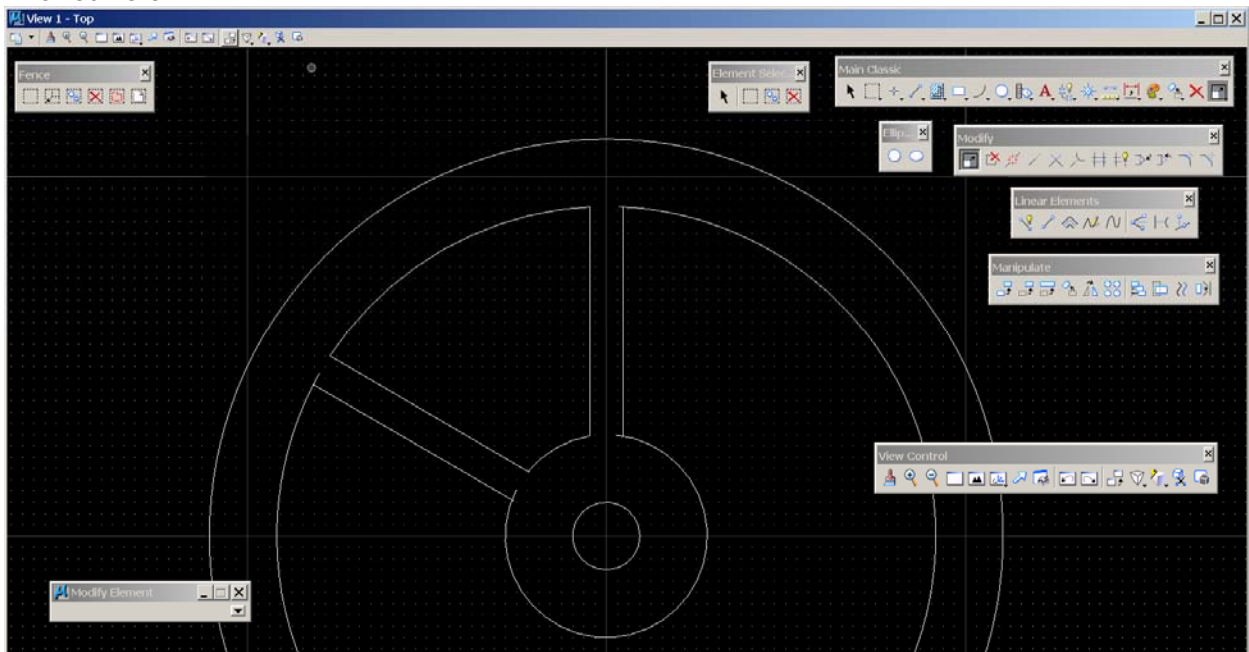


Fig. 10

The next step is to connect the 4 lines for just the section in the northwest part of the wheel. For this task, we use the Extend Elements to Intersection icon again in the Modify Toolbar. Click on the icon, then click on one element to be joined and then the other. You should see the ragged endings of the lines snap together. Do this 4 times for each of the 4 lines. When completed, use the same Partial Delete icon in the Modify Toolbar to delete the remaining parts of the wheel rim and the wheel boss. Delete the vertical line to the right of the completed section and delete the line to the left and below of the completed section. See Figure 11.

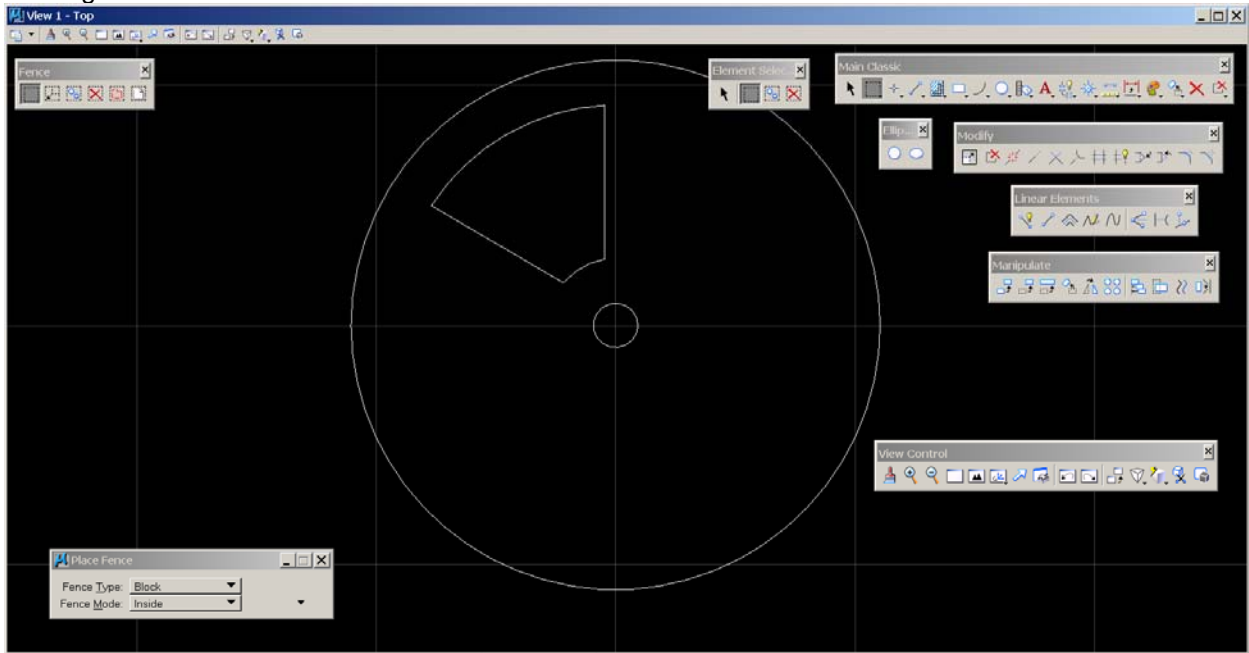


Fig. 11

The last step is to rotate the section using the fence 5 times by 60 degrees to make a total of 6 sections to be cut out by the CNC system. See Figure 12

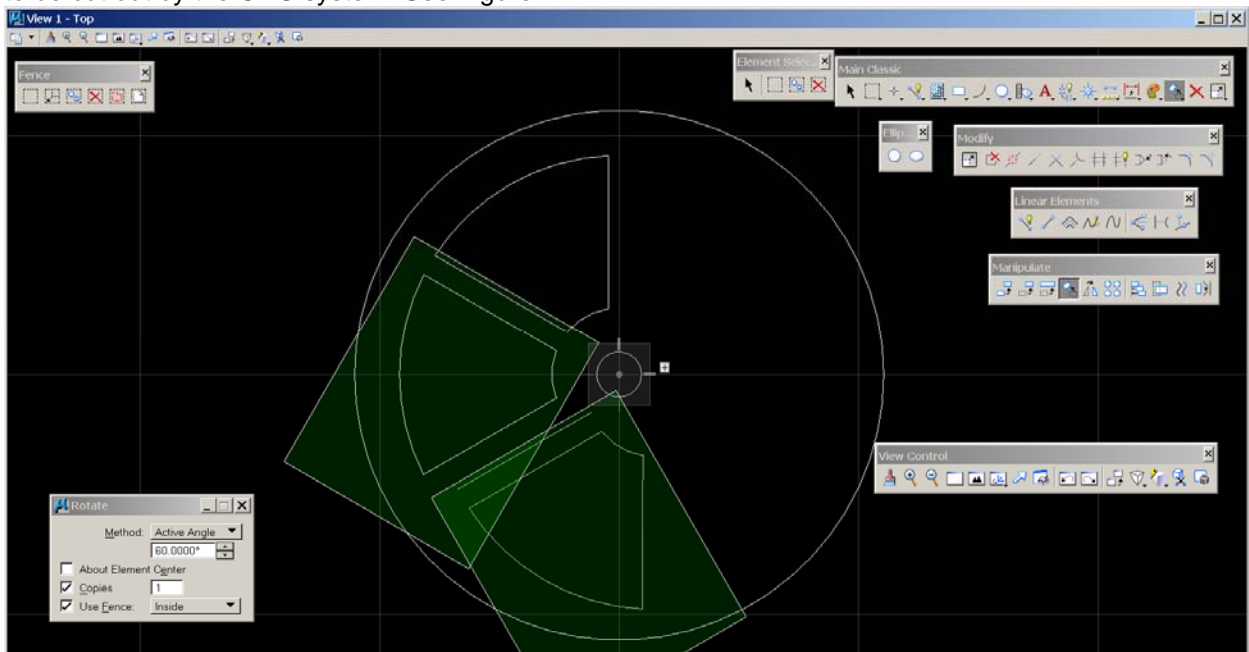


Fig. 12

The completed drawing is shown in Figure 13. For purposes of setting offsets in your CAM program, the I.D. and each section is cut out using an internal offset. We want the cutter path to be inside the lines cutting out on the waste portion. For the O.D. , we want an outside offset so we cut the wheel diameter full size. Assuming you are cutting the wheel out of a plate of sheet brass, the progression of cutting would be the I.D. first, the sections next and finally the O.D. to separate the wheel blank from the plate.

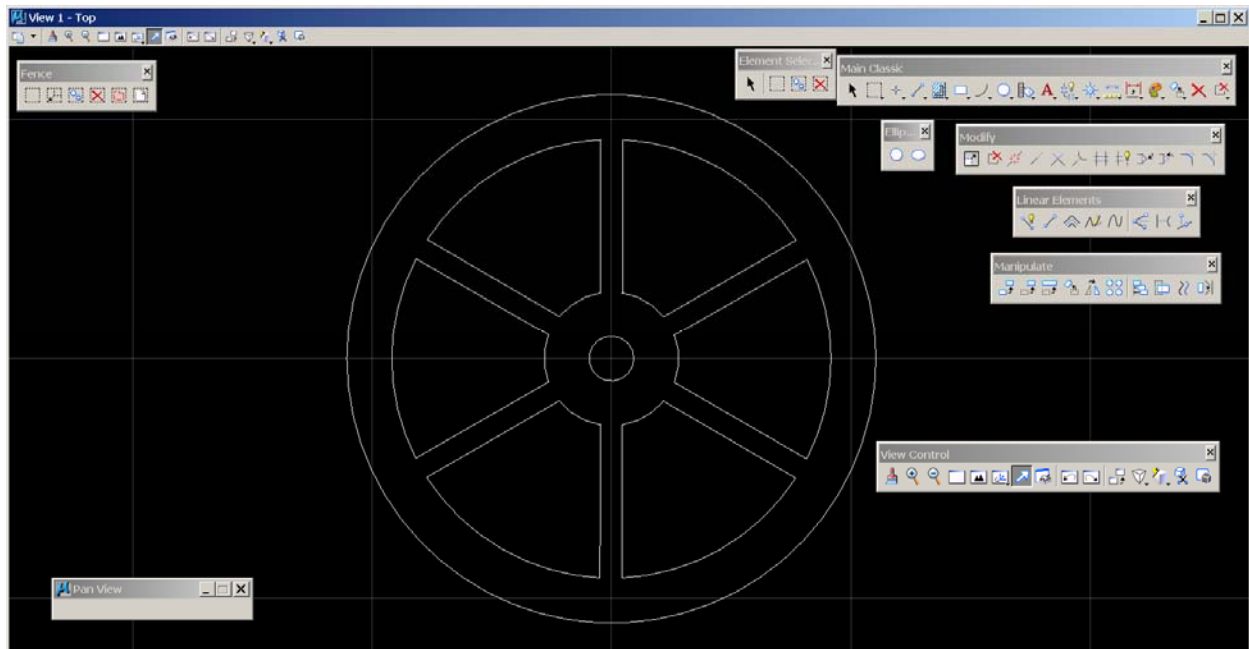


Fig. 13