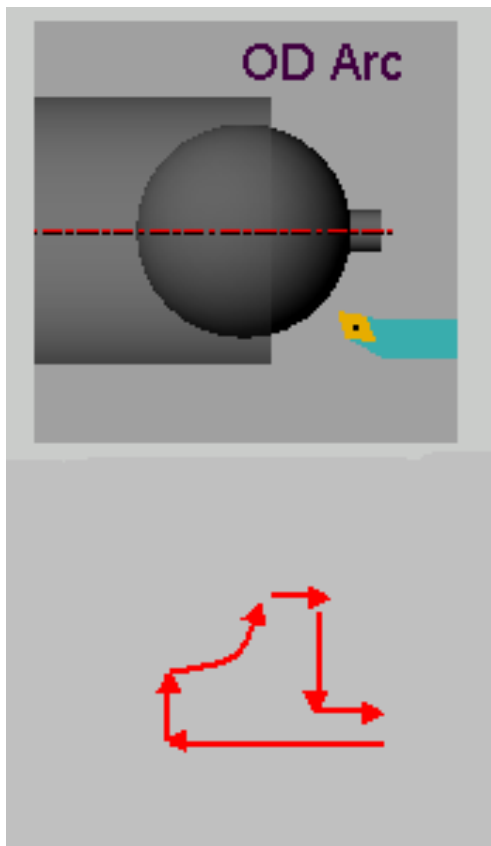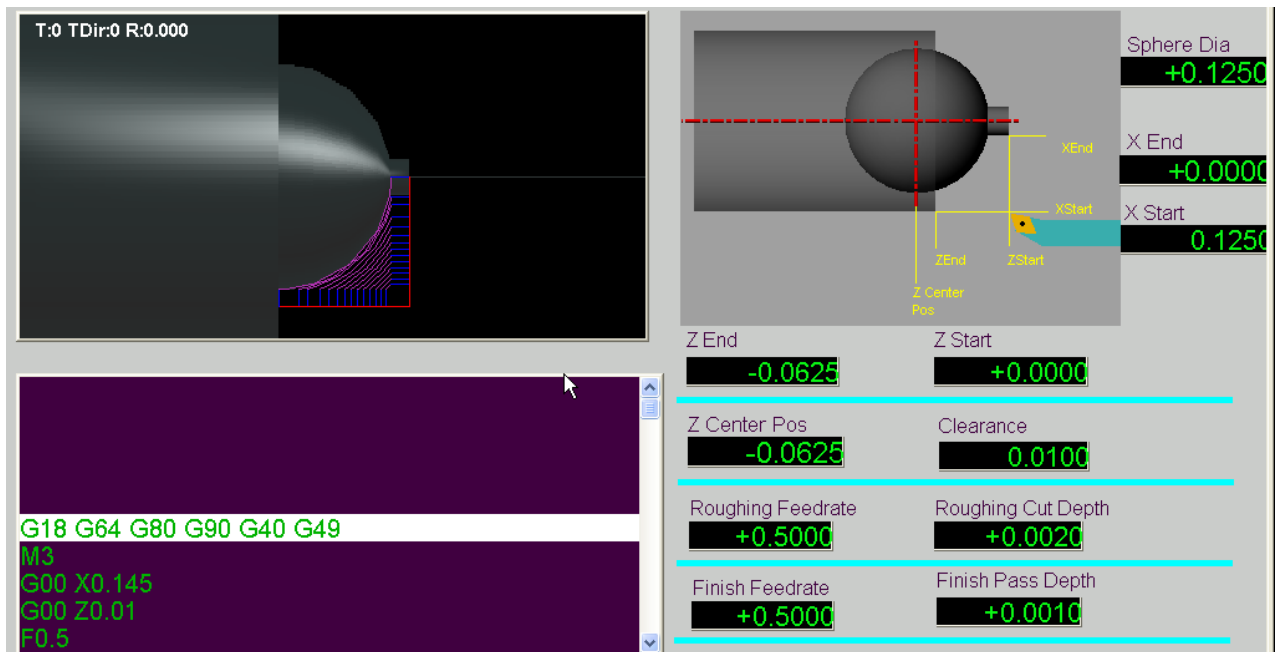**USING OD ARC WIZARD**

The tool pathing is shown below. The cutting path is clockwise and inward towards the center of the shaft as shown below . Finishing of the ball occurs on the last cut. Note that an appropriate tool and correct placement of it is required. A tool, such as one ground for threading, will cut material on the back side when moving about the arc. If it's a small ball and you will need to use a tool ground so that it will not gouge the work. Note that some of the Lathe Tturning Wizards work in radius or diameter mode only, some both. If you get ''crop circles'' after posting code tick Reversed Arc's in front tool post under Turn Options in Configuration > Ports & Pins. You can step through the pathing while in the wizard.

Below is a Mach3 screen shot.

Note the following:

Sphere dia = the diameter of ball you want
XEnd = where you want the cutting to stop
Xstart = the diameter of the ball
Zend = where you want the cutting to stop
Zstart = where you want to start
Z center pos- = where the center of the ball is located (can be positive or negative)
Clearance = is the tools pull out distance along the X &Z axis
The remaining inputs are for defining a rough / finish cut at feedarte.