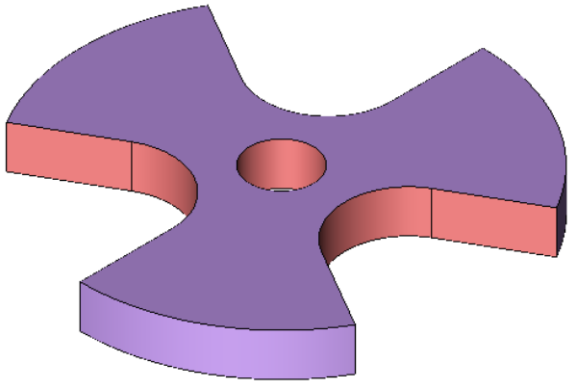
KeyCreator Lesson KC8214

Machining a Three Pocket Yoke (Using Rotate Tool Path)



In this exercise we're going to create the tool paths to machine the three pocket yoke illustrated to the right.

We'll make an assumption that the raw stock is a 10 inch diameter round bar that has been turned down to a precise 9.75 diameter in a lathe prior to the machining operation that creates the lobes.

The end of the bar was also faced off precisely in the lathe and the center hole was machined on the lathe as well. The part was then parted at a rough height of 1.125 inches.

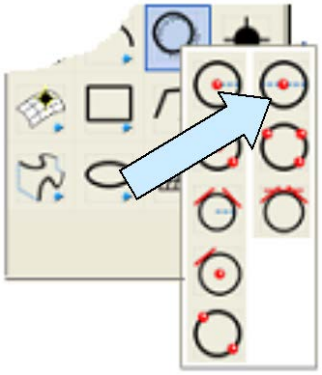
That leaves two operations for our NC milling exercise. (We'll mount the blank turned piece finished face down on our milling machine.)

1: Face off the top surface so the part is an exact 1.000 inch thick.

2: Machine the three yoke pockets.

Creating a Model of the Part

Let's start with a new file in View 1. (The Top View.)



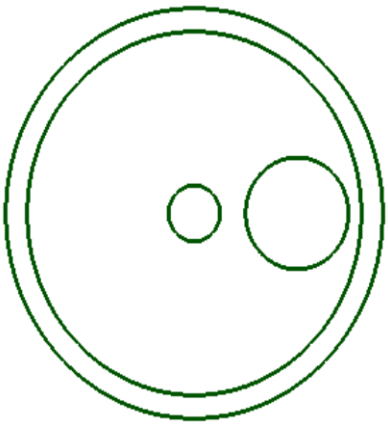
Click on the CREATE CIRCLE BY DIAMETER Icon.
Type 9.75 for the Diameter. Use the KeyIn Option and hit the ENTER Key three times to place the origin.

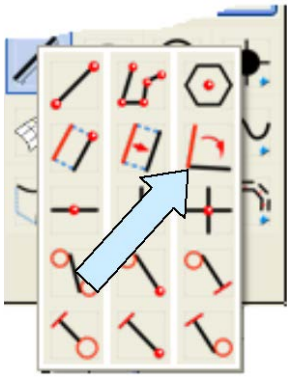
Click on the BACKUP Button.
Type 1.5 for the Diameter. Use the KeyIn Option and hit the ENTER Key three times to place the origin.

Click on the BACKUP Button.
Type 11 for the Diameter. Use the KeyIn Option and hit the ENTER Key three times to place the origin.

Click on the BACKUP Button.
Type 3 for the Diameter.
Use the KeyIn Option, type 3 for the X value and hit the ENTER Key three times

Your screen should look like this:



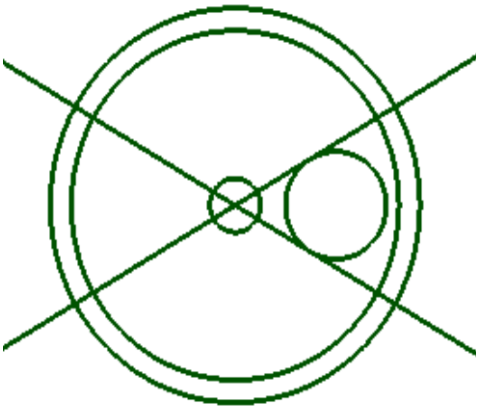


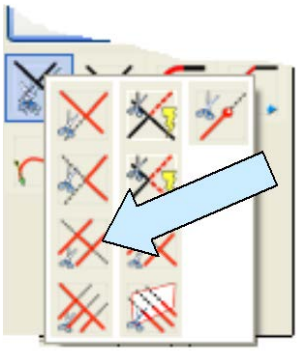
Click on the CREATE LINE AT AN ANGLE Icon.
Type 30 for the angle.

Move the cursor over the 3 inch circle at the 11 O'Clock position and click when the line snaps tangent to the circle.
Click on the BACKUP Button. Type -30 for the angle.

Move the cursor over the 3 inch circle at the 7 O'Clock position and click when the line snaps tangent to the circle.

Your screen should look like this:

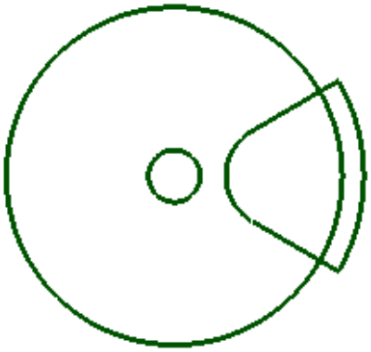




Click on the TRIM DOUBLE Icon.
Click on the 3 inch circle at the 9 O'Clock position and then on the two oblique lines.
Click on the 11 inch circle at the 3 O'Clock position and then on the two oblique lines.

Still using the TRIM DOUBLE Function, click on the upper oblique line between the two arcs you just trimmed and then on each of the arcs. Repeat this process with the lower oblique line.

Your screen should look like this:



Select a new construction color.

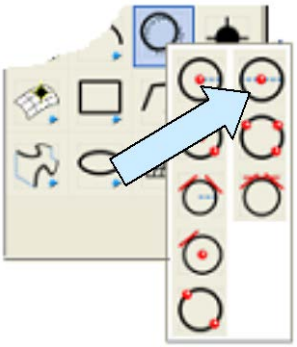
Click on the TOGGLE SPLITTER. Right Mouse Click on the Header Bar and then on the CREATE LEVEL Option in the Menu that appears.

Create a level 2, you can name it “Curves.” Make sure that it is the active level.

Now, click on the XFORM DELTA COPY Icon.

Select the two trimmed arcs and the two oblique lines and hit the ENTER Key.

Hit the ENTER Key three times. (This makes a copy of the selected geometry in the same location.)

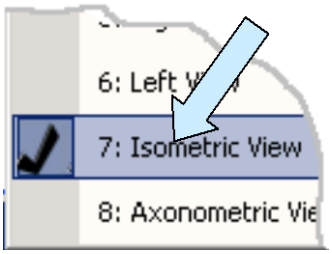


Click on the CREATE CIRCLE BY DIAMETER Icon.
Type 11.5 for the Diameter and hit the ENTER Key.

Hit the ENTER Key three times.

Now, remove level 2 from the display. (We’re going to use the geometry on level 2 for our tool paths later on in this exercise.) Click on Level 1 in the Active Column to make it the active level.

Switch to the Isometric View. (View 7.)

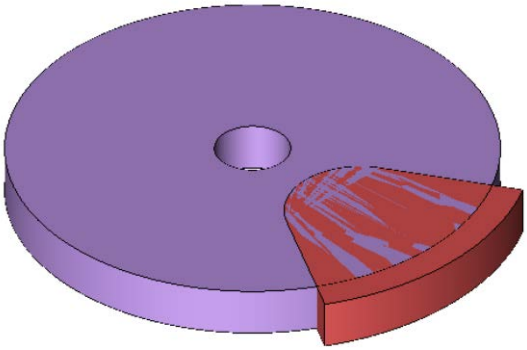


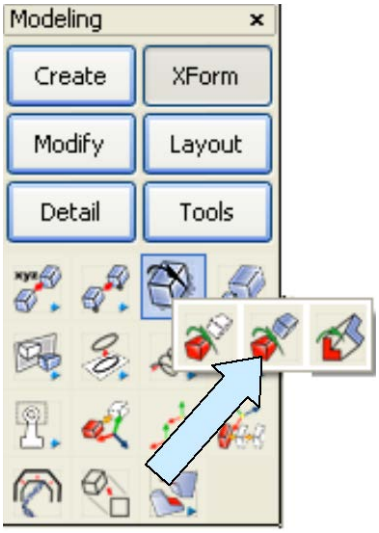
Click on the EXTRUDE Icon.
A Dialog Box appears.
Type 1 for the Length and hit the ENTER Key.

Select the 1.5 diameter circle and the 9.75 diameter circle and hit the ENTER Key. Click on the downward-facing vector.

Next, select the two trimmed arcs and two oblique lines and hit the ENTER Key. Click on the downward-facing vector.

Your screen should look like this:





Click on the XFORM ROTATE COPY Icon.
Select the wedge-shaped solid and hit the ENTER Key.
Type 2 for the Number of Copies.

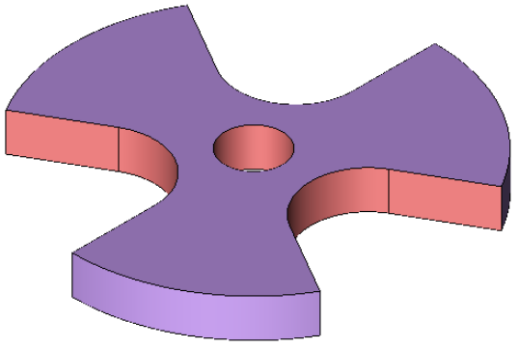
Using the CtrMid Option, click on the top edge of the cylindrical solid and the bottom edge. Type 120 for the Angle.

You will now have three wedge-shaped solids spaced 120 degrees apart.

Click on the BOOLEAN DIFFERENCE Icon.

Click on the cylindrical solid.
Then, select the three wedge-shaped solids.

Your completed part should look like this:

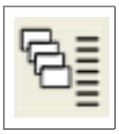


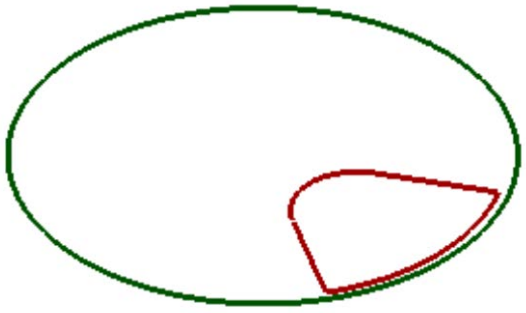
Creating the Tool Paths

I had you go through the creation of the model pictured above as an educational exercise. In an actual shop situation, you really don't need to create this solid model prior to creating the Tool Paths to machine it. We can make do quite nicely with the curves that you placed on level 2 earlier in the exercise.

Take a moment to move the finished part to level 3.

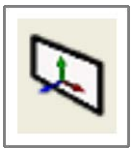
Then, remove level 3 from the display, turn on level 2, and make it the active level.



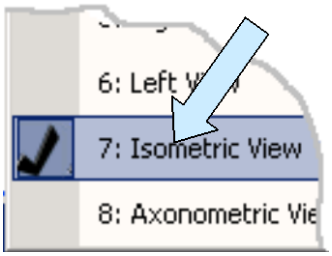


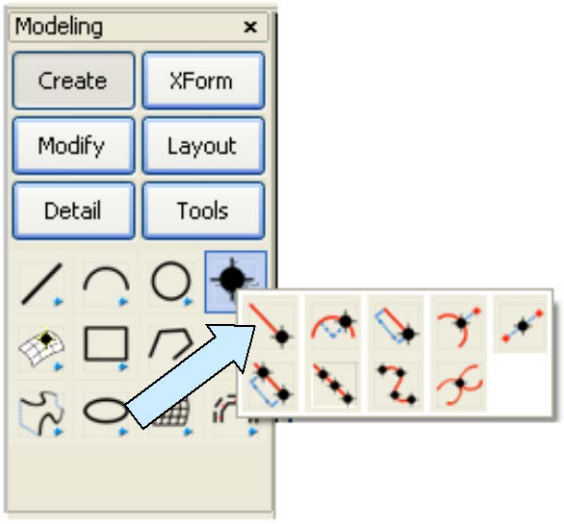
Your screen should look like this:

Click on the CONSTRUCTION PLANE Icon and type 1 for the Plane Number.



Switch to the Isometric View. (View 7.)



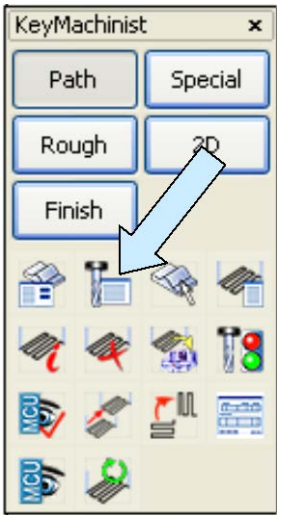


Click on the CREATE POINT AT POSITION Icon.

Use the KeyIn Option. Type –5.5, -5.5, 0 and a point appears to the left of the large circle.

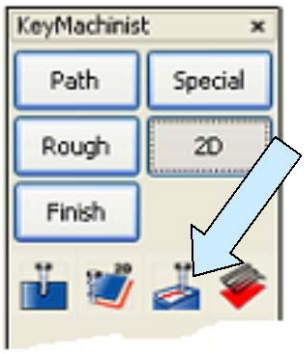
Type 5.5, 5.5, -1.125 and a second point appears to the right of the large circle. These points will represent the corners of the machining volume for verifying the tool paths.

We'll use a 1 inch diameter flat end mill for both machining operations. Take a moment to click on the TOOL LIST Icon.



A Dialog Box appears. Click on the Define New Button at the bottom of the Dialog Box.

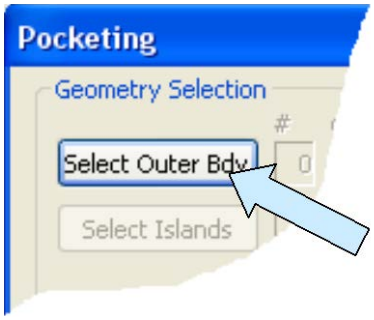
The Add a New Tool Dialog Box appears.
Type "FlatEndMill1" for the name, give it a Tool Number, select the Flat End Mill Option, and type 1.000 for the diameter. Click on the OK Button.



You are returned to the previous Dialog Box. Click on the DONE Button.

Click on the 2D POCKET Icon.

The first sheet of the Pocketing Dialog Box appears. Since we already have our tool selected, click on the Next Button.



The second sheet of the Pocketing Dialog Box appears. Click on the Select Outer Body Button.

A small Dialog Box appears. Click on the Single/Multiple Cavities Having the Same Depth Option and hit the ENTER Key.

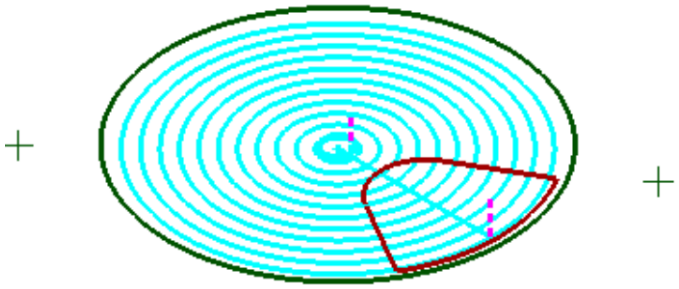
Select the Single Option, click on the large circle on your screen, and hit the ENTER Key twice.

The third sheet of the Pocketing Dialog Box appears. Use the Expand Inside Outward Option and click on the NEXT Button.

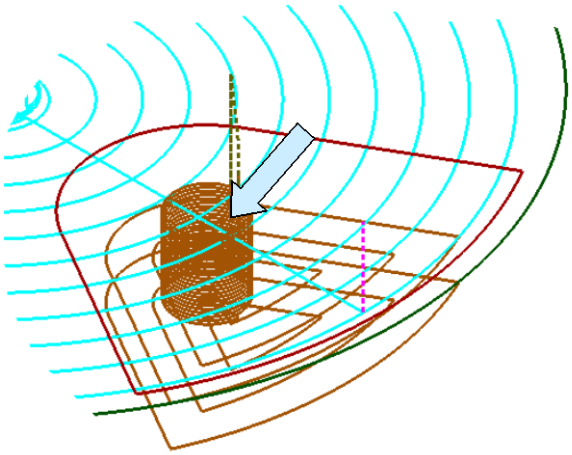
Type –0.125 for the Z Floor value, select 1 for the number of passes, and click on the CREATE PATH Button.

A small Dialog Box appears. Type “FaceOff” for the tool path name and click on the CREATE Button.

You will now have a tool path that looks like this:

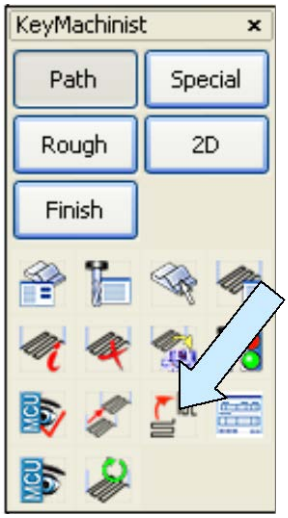


Use the 2D POCKET Function again to create a second tool path.



Use the 1 inch flat end mill and this time select the wedge-shaped profile for the Outer Body. For this tool path, type –1.125 for the Z Floor value and use 2 for the number of passes. Call this path “LobePocket.”

Your tool path should look like this:

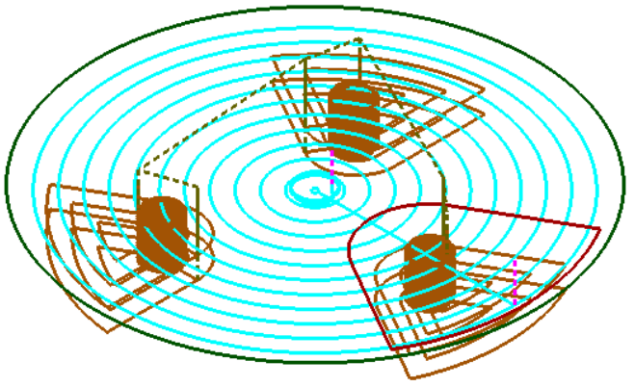


Now, click on the ROTATE TOOL PATH Icon

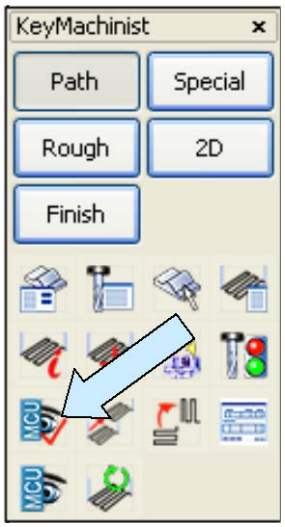
Select the LobePocket tool path and hit the ENTER Key.
Type 2 for the Number of Copies

You are asked if you want to join the tool paths into one.
Select YES.

Type 120 for the Angle.
Using the CtrMid Option, click on the large circle.



Your complex tool path should look like this:



Click on the VERIFY TOOL PATH Icon.

Select the two tool paths and hit the ENTER Key.
Click on the Corners Option and using the Point Option, click on the top point and bottom point that you created earlier.

Then, click on the SKIP Button.

When the Meta Cut Screen appears, hit the PLAY Button to verify the tool path. (Notice that the simulation cuts out of a rectangular volume even though in reality you will be starting with a cylindrical piece of stock.)