

Finding the Center of Rotation on a 5-Axis Table-Table Milling Machine with a Tilt Rotary Unit

In the BobCAD-CAM system when using the multi-axis features your machine must be properly defined. The calculations that must be performed for proper G-Code creation depends on an accurate definition of your machine's kinematics. This document covers how to find the center of rotation for a 5-axis Table-Table machine with a tilt rotary unit so the machine parameters can be correctly defined.

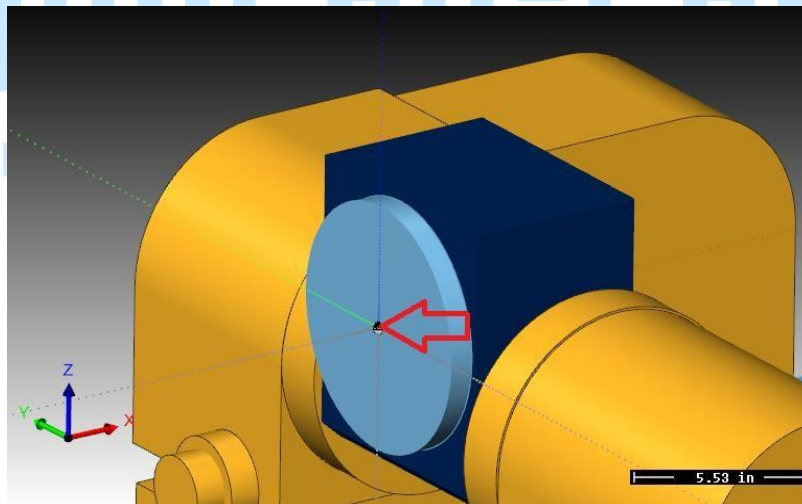
There is a simple test you can use to find the necessary values and we will walk you through this process here in this document.

Step 1 – Locate the X Axis and Z Axis Positions of the 1st Point

The 1st point we need to locate is at the center of the face of the rotary platter. The following image shows the location of the point we are looking for.

Important Note: This document shows a tilt/rotary unit that when at Zero-Zero (both rotation angles at zero), location has the tilting axis sitting where the rotary is aligned to rotate around the X axis. If your zeroing your assembly at a different location, simply apply what you see here to your orientation.

To find this location on your machine, use a tool to touch the face of the platter to find your X axis component. Use the bottom of the tool to touch off the edge of the rotary platter then account for the radius of the platter to get the point at the center.



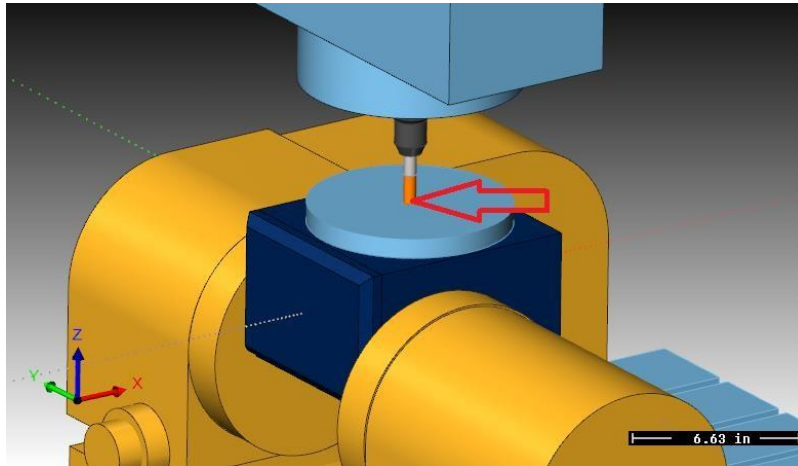
Note the X axis and Z axis positions of this point. (X_1 , Z_1).

X_1 = _____

Z_1 = _____

Step 2 – Locate the X Axis and Z Axis Positions of the 2nd Point

To find the 2nd point you need to rotate the tilting axis to a position that will hold the rotary platter, so it is parallel to the Z axis of the machine tool. The following image shows the location we are looking for.



To find this location on your machine use a tool to touch off the edge of the platter or use an indicator to tram in the XY center of the platter. Use the bottom of the tool to find the face of the rotary platter.

Note the X axis and Z axis positions of this point. (X_2, Z_2).

$X_2 =$ _____

$Z_2 =$ _____

Step 3 – Locate the X and Z Positions of the Intersection Point (X_P & Z_P)

In this step we will use BobCAD-CAM to find the location of the rotation center for your tilt rotary unit. To do this we will plot the two points on the screen and construct geometry to locate the center of rotation.

We are going to locate the 1st point in the BobCAD system at (0, 0) on the screen and the 2nd point will be relative to the 1st point. To do this we need to subtract the values of the two points, so we end up with the difference from the 1st point to the 2nd (X_2' and Z_2').

$X_2' = X_1 - X_2 =$ _____

$Z_2' = Z_1 - Z_2 =$ _____

Now in BobCAD you need to create the 1st point. Create a point at ($X = 0, Y = 0, Z = 0$) on the screen. Next create the 2nd point using X_2' and Z_2' .

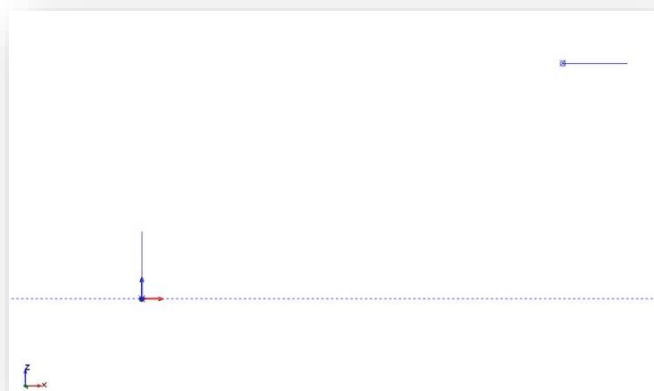


These two points represent the same location on your tilt rotary unit when the unit is in the two different positions. The image below shows an example of what you should have on your screen looking from a FRONT view.

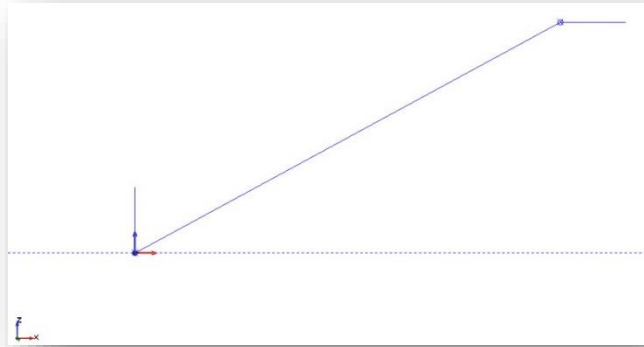
We need to create some reference geometry for the two different orientations. The objective here is to create geometry on the 2nd point that represents the same relative location as it is on the 1st point. The following steps will walk you through this process

- A. Point 1 in the TOP UCS – Create a line from the 1st point location up in Z that is 1.000 long.
Endpoint Data:
X0. Y0. Z0. (1st Point Location)
to X0. Y0. Z1.
- B. Point 2 In the TOP UCS – Create a line from the 2nd point location over in X that is 1.000 long.
Use the Create\Line\Angle option to create this line. Set the Angle to 0 and click the 2nd point as the start point.

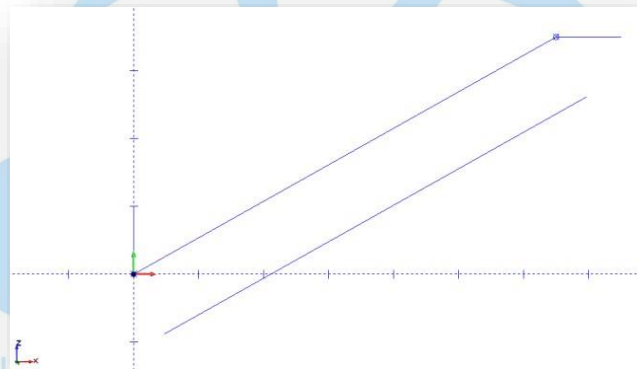
The result of creating this geometry should be two lines. Each line starts on one of the points and the other endpoint represents same point for the tilt rotary in the 2nd position. The following image shows the result of this step.



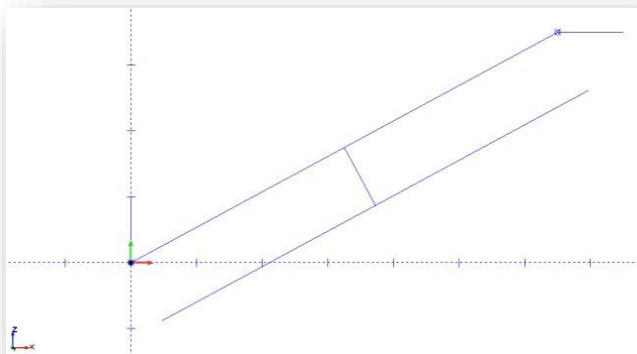
Now we connect the two ends of our reference geometry to each other by connecting the relating endpoints to each other. Use the Create\Line\Continuous command to do this. Next create a perpendicular line in the center of each of the two lines joining the reference geometry. Create reference line from point 1 to point 2 as it is shown in the figure below.



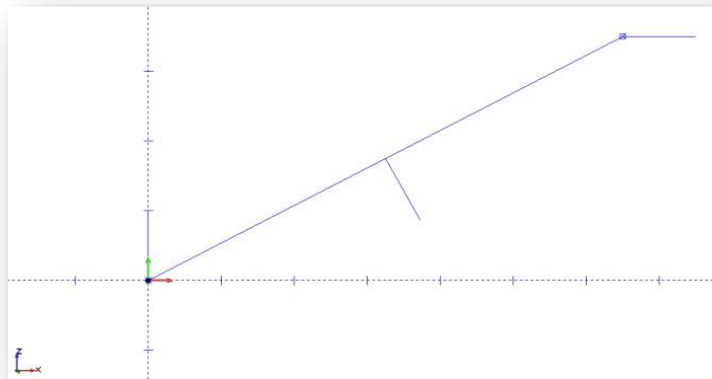
Now we create construction geometry used to construct a perpendicular line. Set the USC to FRONT. Next offset the join line by 1.000 down using the Create\Line\Parallel command. The following image shows the result of this step.



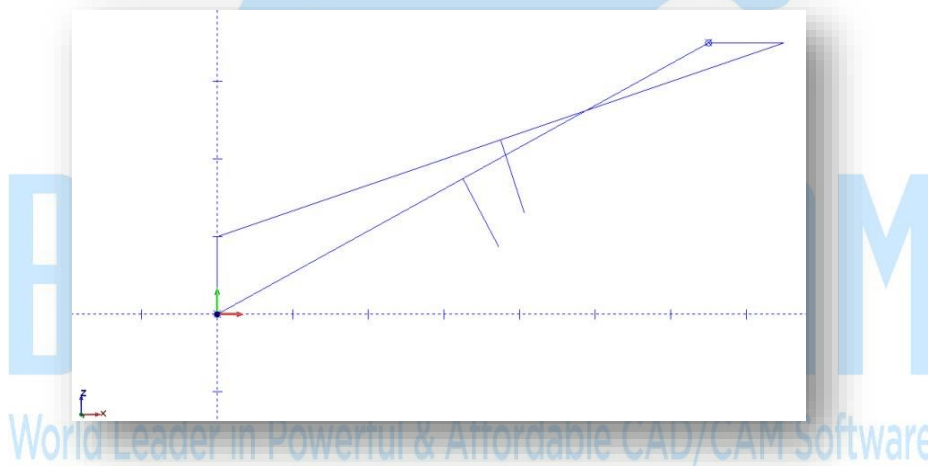
Next create a line from the mid-point of the 1st join line to the construction line just created. The following image shows the result of this step.



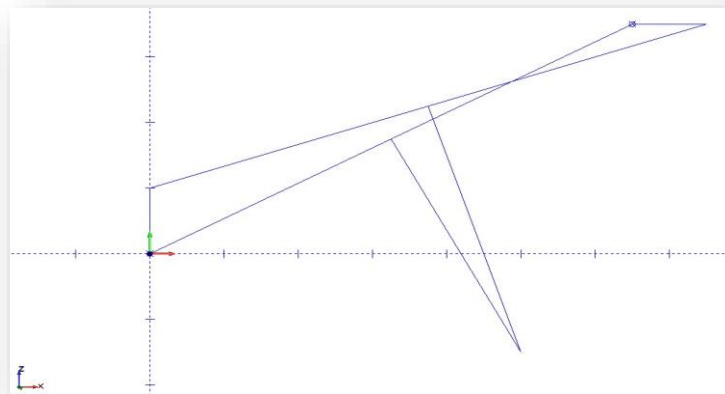
Now Delete the construction line as you no long need it. The following image shows the result of this step.



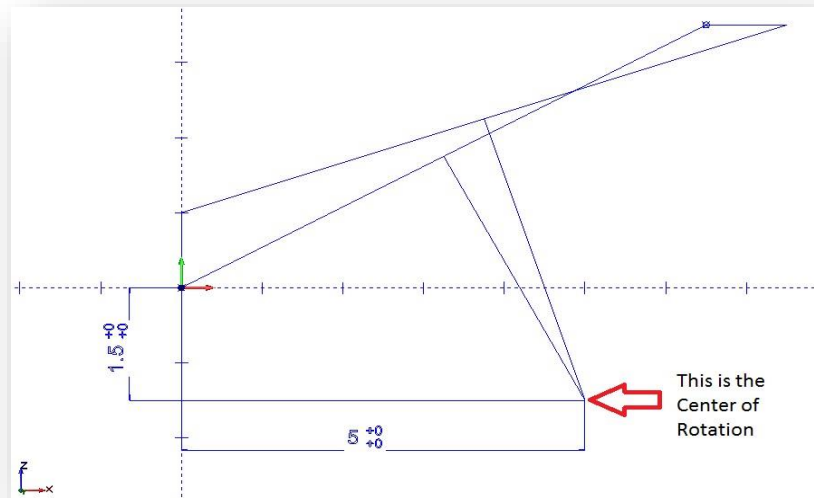
Now repeat the same steps to create the join line and perpendicular center line for the 2nd endpoints of our reference geometry. The following image shows the result of these steps.



Use the Trim Two option to trim the two perpendicular lines to each other. The intersection point of these two lines is the center of rotation. The following image shows the result of these steps.



Using the dimension commands, you can locate the location of the center of rotation relative to the 1st point.



This point represents the X and Z coordinates of the center of rotation for the tilting axis.

Note these center point values (they are relative to 1st point)

X_p = _____

Z_p = _____

