

Setting up a right angle head in Mastercam

It is important to have a good understanding of work planes and tool planes.

Always think of yourself as the end of the tool looking towards the work piece.

Which way is X+?

Which way is Y+?

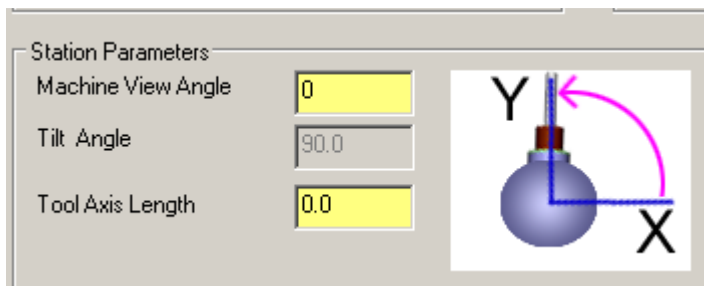
Which way is Z+?

There really isn't any difference between a Horizontal and Vertical machine: X+ is always to your right, Y+ is up and Z+ is towards you.

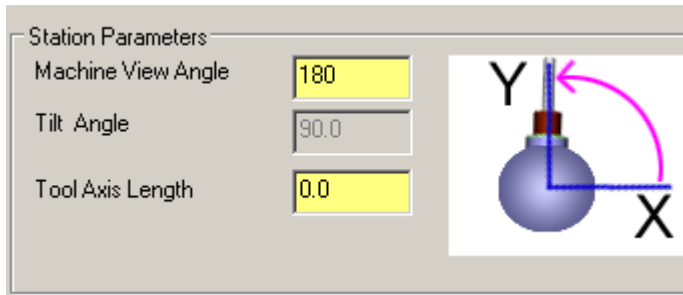
One of the things that needs to be done is; setting the correct orientation for a right angle head in the machine definition.

Basically you have 4 choices:

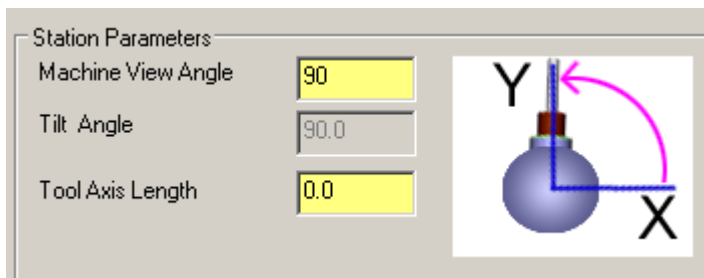
X tool to the right:



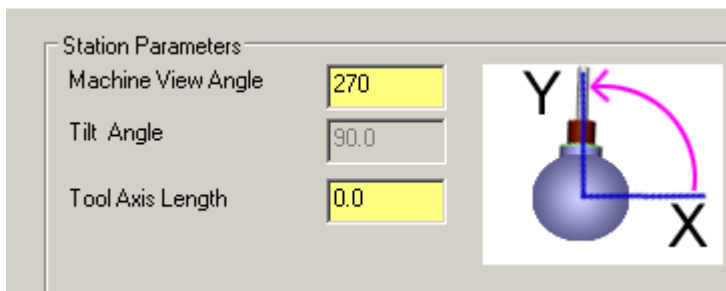
X tool to the left



Y tool up



Y tool down

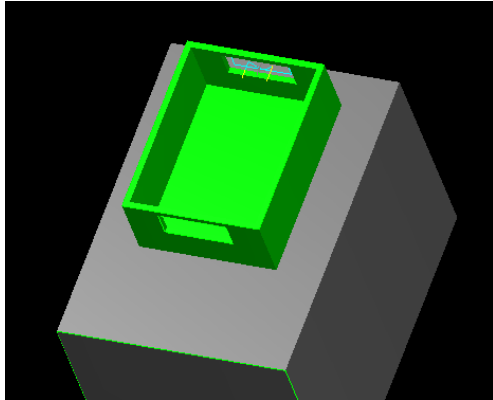


Change the Machine View Angle according to the way that you have mounted the RAH.

Generating a toolpath

Change the Plane in Mastercam to match the plane that you want to machine.

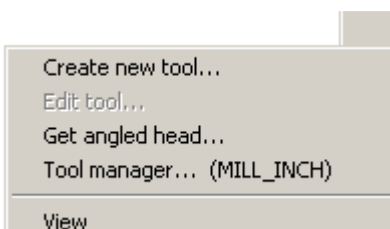
In my example I am going to contour the inside top edge of a rectangle in a box



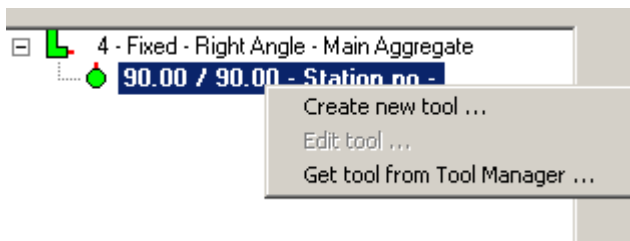
(The gray block is a tombstone.)

I changed the plane to Front and then selected the geometry for the contour.

In the contour toolpath operation I right mouse clicked on the Get angled head.



Then I right clicked on the "Station no"



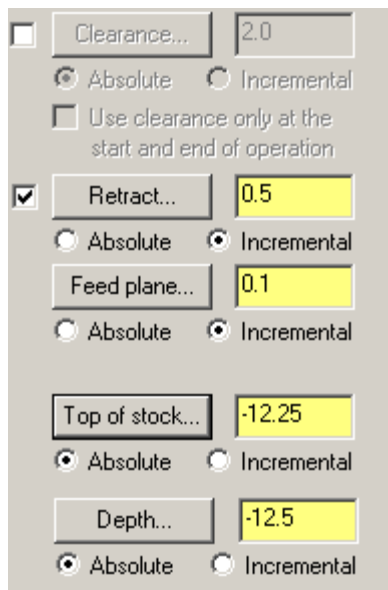
Then “Get tool from Tool Manager”

I can now select a tool as I normally would. Or if a tool was already selected, I can edit it.

Setting cut depths

Pay careful attention to the settings for the cut depths

Here is the way I set the depths for my example:



If you don't understand how the absolute / incremental values work in Mastercam, I suggest that you learn. They are very powerful tools. They are NOT the way we use Absolute and Incremental at the machine.

Briefly:

Absolute and Incremental in Mastercam are in relation to location of the geometry that was selected for a toolpath.

In the example above, the geometry that I selected was at -12.5. If I change the check box to incremental, I would want to set the value to zero. If I put 1.0 in the depth box and had incremental set, the machine would move to -11.5 (the depth of the geometry selected plus the 1.0)

With incremental, everything stacks bottom to top in the depth fields.
(Depth on the bottom and Clearance on the top)

What this means is: if you make one of the fields incremental and enter a value, the machine will move to that value above the next lower field.

As an example: I normally keep the Feed Plane at Incremental and set to .1

That way, no matter where the top of stock is set, my feed (rapid) plane will be .1 above the top of stock.

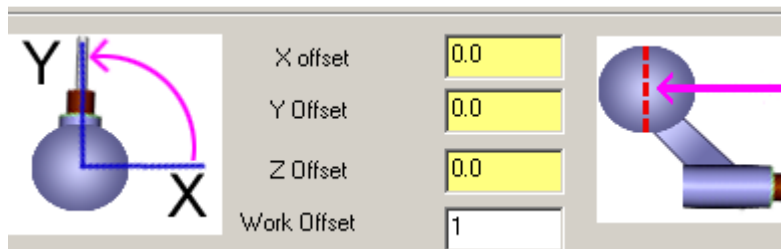
If I set the Retract to incremental at .5 as in the example, the retract height will be .5 above the next lower field (feed plane) so the machine will retract to -11.65 in absolute values. -12.25 (top of stock) $+ .1$ (feed plane) $+ .5$ Retract height)

Compensating for the tool length.

There are 2 ways of compensating for the length of the tool. (Remember that the machine spindle is pointing towards the part, BUT we need to compensate for the length of the tool plus the right angle head.)

#1.

The first way is to set up a Work Offset on the machine. Add the pivot length (the center of the machine spindle to the tip of the tool) to the Y axis for whatever work offset that was specified when you set up the aggregate head in the machine definition:



If you put 1 in the work offset field you will get a G55 when you post, 2 will give you a G56 and so on.

The benefit of doing the tool length compensation this way is you can have vertical tools and right angle tools all in the same program and do tool changes if your machine will allow it.

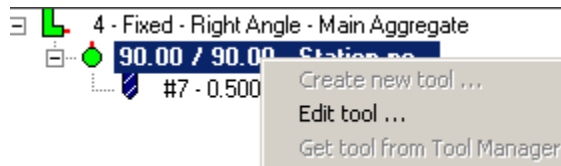
In the post is a “switch” that will allow you to turn this feature on and off.

Open the aggregate.pst file in an editor and search for use_ra_offs. If it is yes, it will output the work offset that is stored in the flex aggregate in the machine definition. If it is no, it will output the work offset from the planes dialog box in the operation.

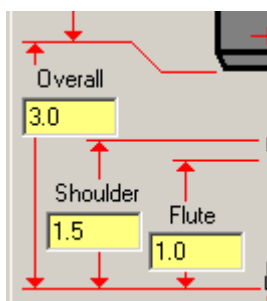
#2.

The second way is to set the overall length in the tool definition:

Right mouse click on the “Station no” again and select Edit tool



Take a measurement from the center of the machine spindle (or pivot point) to the tip of the tool. Enter that number in the overall field in the tool definition:



This value will be added to the Y axis moves during posting.

This is also a “switch” in the post.

Open the post in an editor and search for use_oal

```
use_oal      : yes$      #Scale coordinates to tool length with Aggregates  
use_g8      : yes$      #Use G8 P1 for Fast corner (adv preview) function?  
use_da      : no$       #Output codes to raise and lower dust cover?
```

Change it to yes if it is no.