GIBBSCAM® 14

Version 14 : September 2020

Advanced CS Tutorials

CAMBRIO

Proprietary Notice

CAMBRIO
Modified: Thursday, March 11, 2021 6:42 PM

ADVANCED CS TUTORIALS

Using Coordinate Systems

Introduction

The following tutorials will demonstrate how to work with coordinate systems or “planes.” There are three primary coordinate systems you should become familiar with when defining parts, the XY, XZ and YZ coordinate systems.

You should already be familiar with the XY plane. The X axis is horizontal, the Y axis is vertical and the Z axis is the depth axis. The depth axis of a plane is not visible but its orientation can be seen by a “+” or “-” at the origin of the plane. The plus or minus symbol represents whether we are looking at the positive or negative side of the depth axis.

All standard milling parts are defined from the XY plane (which cannot be modified). The XZ and YZ axes are defined by rotating 90° clockwise about the X or Y axis. From these coordinate systems we can define other coordinate systems that are rotated, flipped or modified at an angle.

The first tutorial begins with a general overview of different concepts when working with Coordinate systems. Then, we will explore adding geometry to modified Coordinate Systems.

Terms

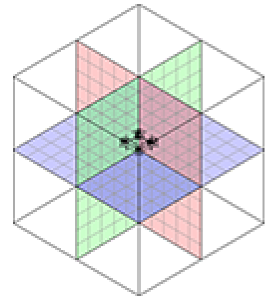
CS
The term CS means “coordinate system” and is used frequently.

CS1, CS2, etc.:
CS followed by a number denotes a particular CS in the CS list. All coordinate systems have a number and can be named.

mm
All parts in these tutorials are in metric values. A millimeter is about 1/25th of an inch. The part files generally have fairly round numbers so the mental conversion to inch should be fairly easy, i.e. 100mm is about 4".

Coordinate System Basics

This exercise is designed to introduce you to creating coordinate systems and to familiarize you with the interface items used. The exercise has two parts. The first part is simply creating the standard coordinate systems (the XY, XZ and YZ planes) at the origin of the part. In the second part of the exercise we will learn another method for making coordinate systems as well as get an introduction on modifying the planes.



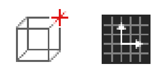
Creating the Primary Planes

The XY Plane

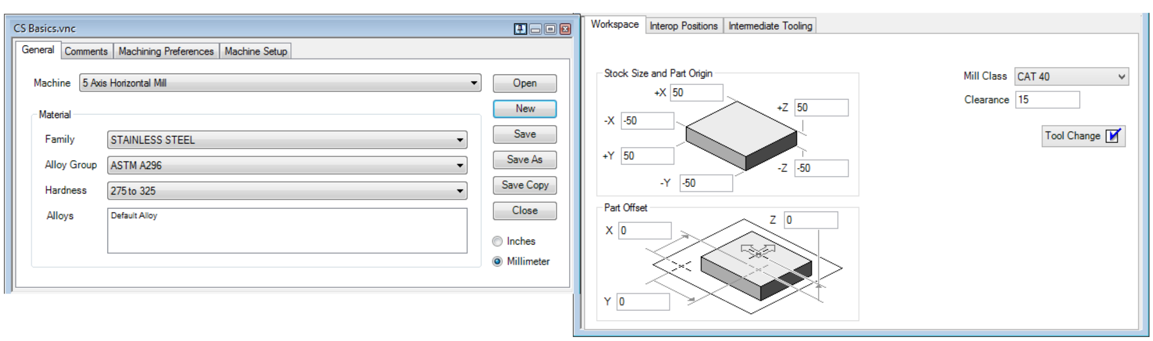
We will start by creating a 100mm cube. You do not need to select a specific MDD as we will not be creating any toolpath for this part but you should select a vertical mill MDD to ensure your results look the same as those in the tutorial. We have selected a 5 Axis Vertical Mill MDD for this example as that is what would be required to access all the coordinate systems.

1. Launch GibbsCAM if it is not running.

2. Ensure that the Show Stock & Origin and Show CS items are active.

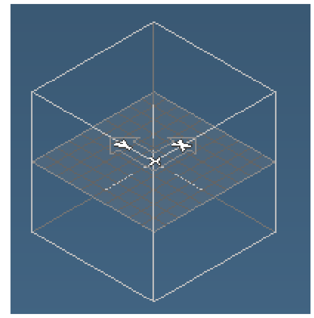


3. Create a new part named CS Basics with dimensions as shown below.



The system automatically creates the XY plane when a new part is made. This base coordinate system may not be modified. The position of the XY plane is defined by the origin of the part, as set up in the Document Control dialog. We placed the origin at the very center of the 100mm cube.

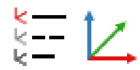
4. Switch to the isometric view (Ctrl+I) and unzoom (Ctrl+U) the part.

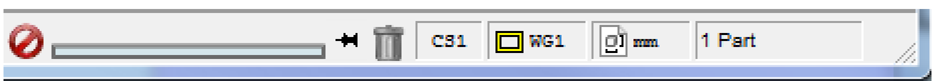


You can now clearly see the origin and that the XY plane is at the center of the part. The + at the center of the axes markers indicates that we are looking at the positive end of the depth axis. A "–" would indicate the negative view.

Creating the XZ and YZ Planes

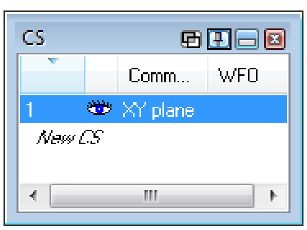
1. Open the CS list and Coordinate System palette by clicking on the Coordinate Systems and CS palette buttons in the Main palette, or by double-clicking the CS button on the status bar at the bottom of the screen.

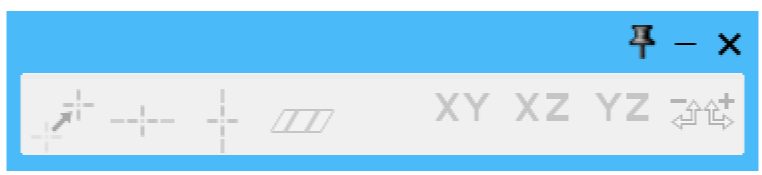




The CS list is used to create, show and switch between coordinate systems. The CS palette is used to modify the current coordinate system. The name of the current coordinate system is displayed in the CS palette's title bar.

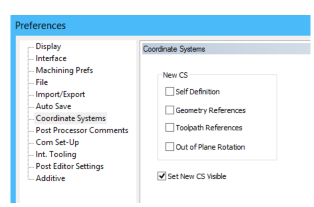
When you open these items you will only see the XY plane in the CS list. All of the buttons in the CS palette are disabled or grayed out. This is because the XY plane cannot be modified.



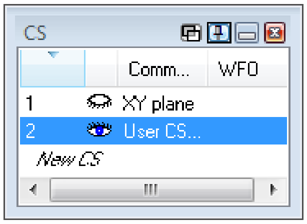


CS List CS Palette

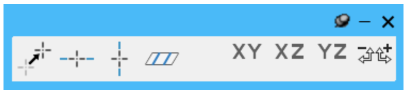
2. Go to File>Preferences>Coordinate Systems. Ensure the settings as shown right are selected.



3. Click New CS in the CS list to create a new coordinate system.

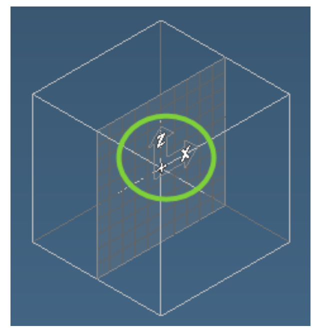


The new coordinate system is called User CS. The new coordinate system is in same place as the XY plane. By default, newly created coordinate systems are based on the last active coordinate system.



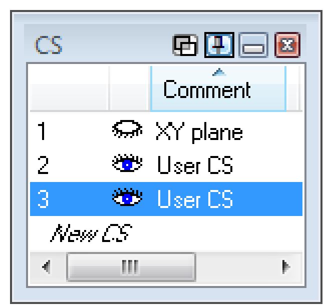
Now that we have a coordinate system that can be modified, the CS palette is active and can be used to modify the coordinate system.

4. Click the XZ button in the CS palette.



The coordinate system has been repositioned. At the origin of the CS you can see that the horizontal axis is marked as X and the vertical axis is marked as Z. We will now make the YZ plane.

5. Click New CS to make a second coordinate system.



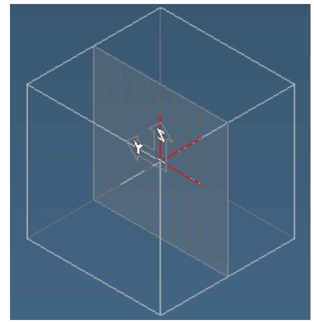
The new coordinate system is called User CS and it is based on the XZ plane. Each new CS will continue to increment the name in the list.

There are three short red lines at the center of the part. These lines are a marker for the origin of a visible coordinate system. A visible CS has an open eye in the CS list. When a CS is visible and not active the grid is not displayed.

We will now modify the current CS to be the YZ plane.

6. Click the YZ button in the CS palette.

The coordinate system has been repositioned. At the origin of the CS you can see that the horizontal axis is marked as Y and the vertical axis is marked as Z. The Z is backwards indicating that we are looking at the negative side of the depth axis.



We have just created the three primary planes of a mill part. The origin of each CS is at the center of the part. We will now modify this part to make it a little more useful by defining the coordinate systems at the edge of our stock boundary. While doing this we will learn how to name coordinate systems and get an introduction to moving them around.

Modifying the Properties of Coordinate Systems

Naming Coordinate Systems

It is highly recommended that you give coordinate systems a name or description. This is especially true when working with parts that contain more than a few coordinate systems.

1. With CS3 highlighted `Double-click` the name field.

   The name field becomes a text entry field.

2. Rename CS3 `YZ plane`.
3. Rename CS2 `XZ plane`.
4. Select CS3 YZ plane.

Displaying and Hiding Coordinate Systems

There is an eye icon next to each coordinate system in the CS list. The eye works just like the workgroup list. When the eye is open you can see a CS, when closed the CS is hidden. The CS list is different from the Workgroup list in that hiding a workgroup will hide the geometry that is in the workgroup. Hiding a coordinate system does not hide any geometry associated with the CS. Hiding a CS merely hides the red coordinate system marker that designates the origin of a CS.

1. `Double-click` the eye icon next to CS2 (the XZ plane) to hide the coordinate system.

   Did you notice how the red coordinate system marker disappeared?

2. `Double-click` the closed eye icon next to CS2 to show the CS.

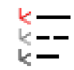
The coordinate system marker is now back.

Switching Coordinate Systems

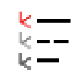
Switching between coordinate systems is very easy and there are multiple ways to change the current CS. The most common method is to use the CS list. Simply `click` anywhere in the CS comment, or number in the dialog box and you will switch to it.

1. `Click` CS2 XZ plane.

Did you notice how the CS palette changed from the YZ plane to the XZ plane? We will now use another method to switch between coordinate systems – the Coordinate Systems button in the Main palette.

2. Close the CS list by `clicking` the close button or the CS list button. 

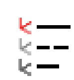
When the CS list is not open the CS list button displays a small black triangle in the bottom left corner of the icon. The triangle signifies that this button has a flyout menu similar to the standard menus such as Edit > Select. Hold the button down to open flyout CS list. This is a quick list to all the coordinate systems defined in the part.

3. `Click` and hold the CS list button until a menu apears. Continue holding the mouse until you have highlighted CS3, the YZ plane. Let go the mouse and the coordinate system becomes active. 

ⓘ There is a third method for switching to a different CS. Clicking on the coordinate system marker of a visible CS will switch you to that CS. Since the origins for the current coordinate systems are virtually identical, it would be difficult to demonstrate this. However, it is discussed in a later step in this tutorial.

Deleting Coordinate Systems

Deleting a coordinate system is as easy as making one. You simply select it and press the delete key on your keyboard. Alternatively, you may click the trash icon at the bottom corner of the workspace. If there is geometry assigned to the CS it may be deleted or changed to CS1.

1. Open the CS list. 

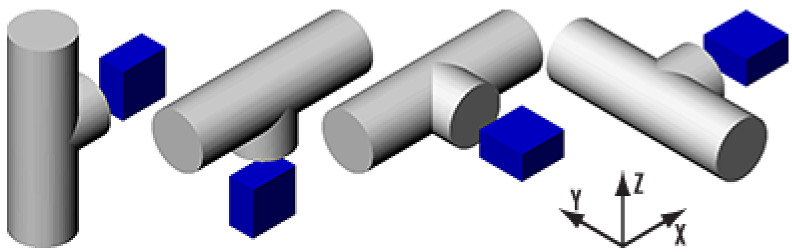
2. Remove CS3 YZ plane, by pressing the Trash button . 

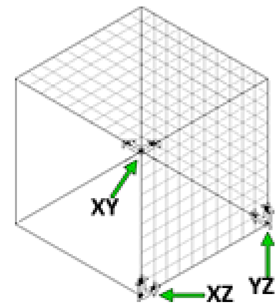
3. Remove CS2 XZ plane, by pressing the Delete key. 

You should have only CS1 in the list. We are now ready to move on to learning how to modify coordinate systems.

Changing the Origin of a Plane

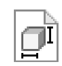
We will now redefine the stock boundaries so the part's primary coordinate systems are on the edge of the stock. We will define the part so that all geometry will be in the H+V+ quadrants. Therefore we want the coordinate system origins on the bottom left corner of the part's sides. This setup is useful for 5-axis machines that can access these faces.

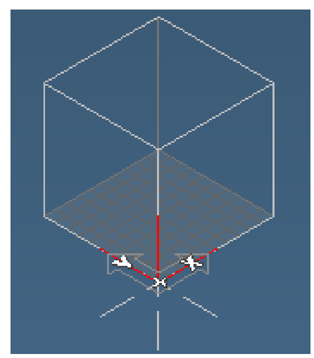




The XY Plane

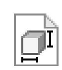
1. Open the Document Control dialog and set the dimensions to be +X= `100`, –X= `0`, +Y= `100`, –Y= `0`, +Z= `100`, –Z= `0`.

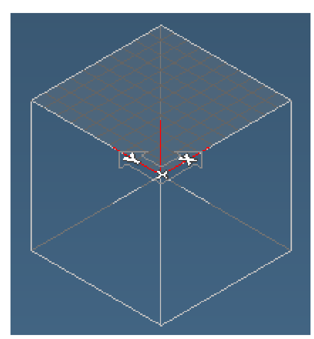




This placed the XY plane at the bottom of the cube, but we want it at the top. Since CS1 cannot be modified, the only way to accomplish this is to change the Document Control dialog settings.

2. Change the Document Control dialog settings to +X= `100`, –X= `0`, +Y= `100`, –Y= `0`, +Z= `0`, –Z= `-100`.





As you can see, the XY plane is now at the top face of the workspace stock with the origin at the bottom left corner of the plane.

Using XYZ and HVD Values

We will first create and name the XZ and YZ planes. These planes will be identical to the XY plane but we will move the coordinate systems into place.

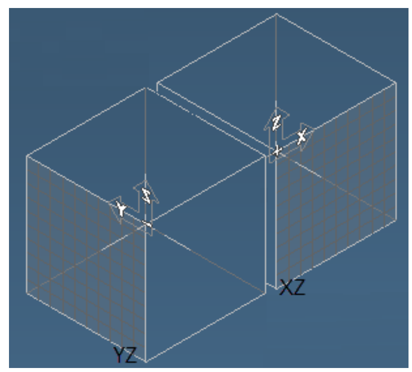
1. Click New CS in the CS list.

> ⓘ Alternatively, you can use the Coordinate Systems button flyout menu to create a new CS, and switch between different coordinate systems. For more information on the menu, see "Switching Coordinate Systems" on page 8.

2. Rename the CS `XZ plane`.
3. Create another new coordinate system and name it `YZ plane`.
4. Select the XZ plane in the CS list and click the XZ button.

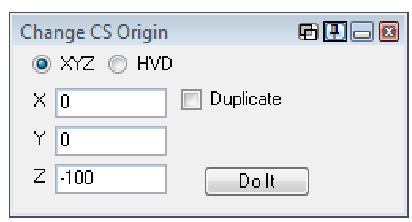
5. Select the YZ plane in the CS list and click the YZ button.

While these coordinate systems are fully defined as the XZ and YZ planes, they still need to be modified to be on the lower left corner of the plane. The origin of the XZ plane must be moved down to the bottom left corner and the YZ plane must be moved across the part and then down.

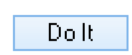


6. Select the XZ plane CS.
7. Click the Change CS Origin button.

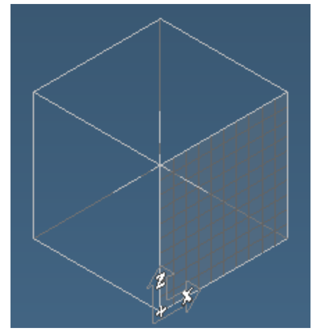
The Change CS Origin dialog allows you to move a coordinate system's origin in ether XYZ (absolute from the XY plane) or HVD (relative) values. We need to move this plane by -100 in Z to get it to the desired location.



8. Select XYZ and enter the data shown and Do it.



The origin for CS2 is now at the bottom left corner of the stock.

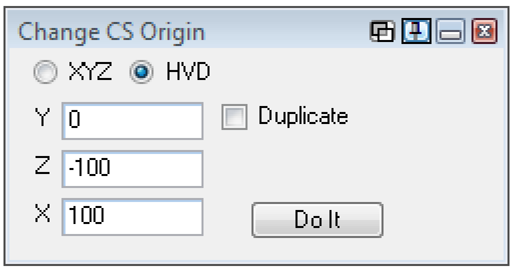


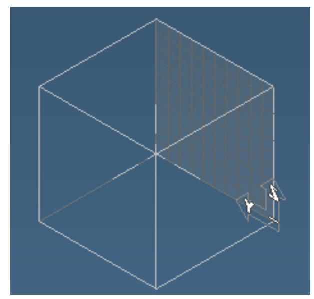
We will now modify the YZ plane.

9. Select the YZ plane by `clicking` the CS marker .

We need to move the plane across the part (100mm) and down (-100mm). Since these values are relative to the current origin, we will use the HVD option.

10. Select HVD and change the data as shown and Do it.



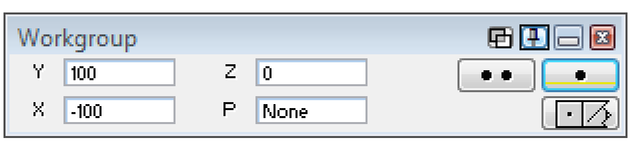


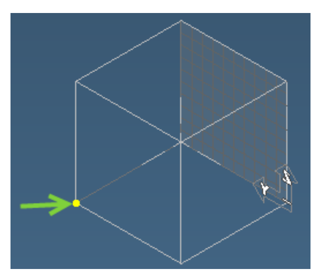
We now have a part set up that has the three primary planes in the lower left corner of the top, front and side of the part. We will now create another CS to illustrate another important function for creating coordinate systems.

Using Existing Geometry

We will now try another useful method for aligning a plane. Typically, a part has an existing point or a known point that can be used as a reference for a plane's origin.

1. In the YZ plane, create the point shown.





2. Create a new CS and name it `YZ plane reversed`.

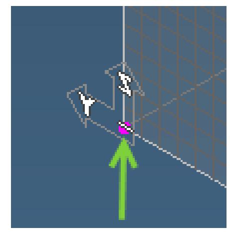
If there is an existing point to use for an origin the point may be used to automatically align the plane.

3. Select the point (which has changed from yellow to pink and will now turn to green).
4. Click the Change CS Origin button .

5. Deselect the point.

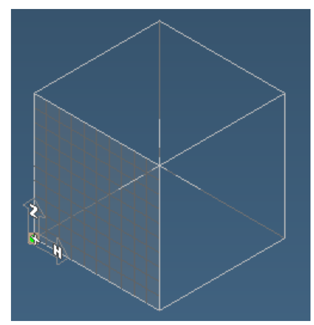
Toggling the Depth Axis

You will notice that the axis markers for YZ plane reversed are displayed in reverse.



We will use the Toggle Depth button to change the polarity of the depth axis. The depth axis is always indicated by a “+” or “-” sign at the intersection of the two axes markers. The “+” indicates that the positive direction of the depth axis in the current view is pointing forward or coming towards you. The “-” indicates that the negative direction of the depth axis is pointing torward you.

1. Click the Toggle Depth button .



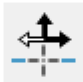
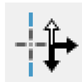
When you toggle the depth, the system will rotate the depth axis 180° around the vertical axis. This has the effect of inverting or flipping the direction of the horizontal axis as well as the depth axis. The vertical axis is not affected. This only has an effect on the orientation of the coordinate system not on the actual orientation of the part.

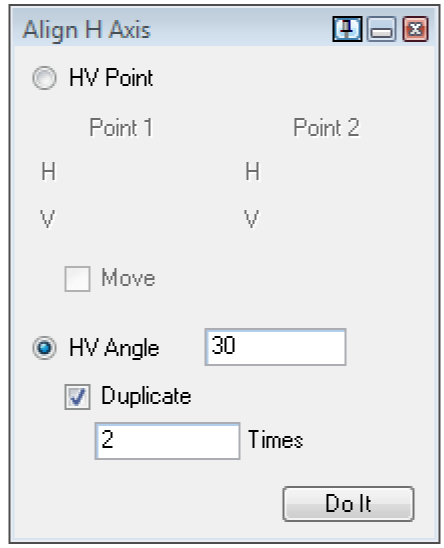
Aligning Coordinate Systems

There are many ways to align a coordinate system. Coordinate Systems can be aligned by designating where a horizontal or vertical axis lies; planes can be snapped to geometry; and planes can be rotated. We will now cover four primary methods of aligning a CS. the first two items will use the Align Horizontal Axis and Align Vertical Axis buttons. We will then use the Align CS button for two other methods.

Single Axis Alignment by Manual Input

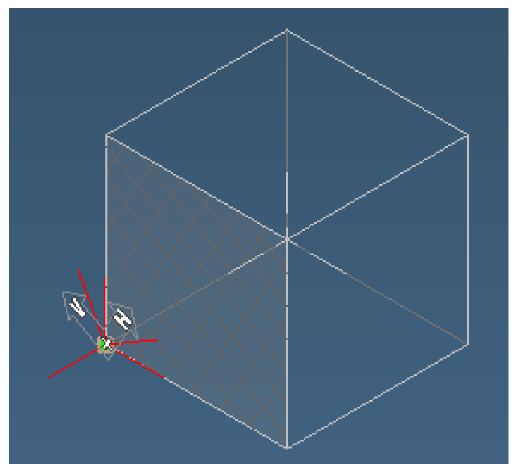
1. Create a new CS based on CS4 (Highlight 4 and click New CS) and name it Learning.
2. Ensure that all geometry is deselected and click the Align H Axis button.

The  Align H Axis (or  V Axis) dialog allows us to align the horizontal or vertical axis of the current CS by using either user input points or to rotate the plane at a given angle. By default, actions performed by this function will duplicate and modify the current plane instead of moving it.



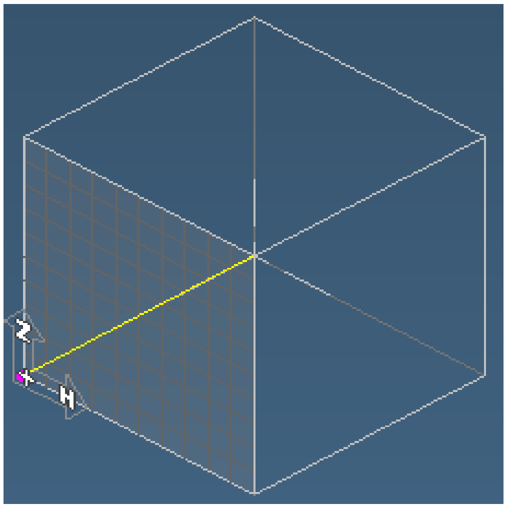
3. Enter the values shown and Do It.

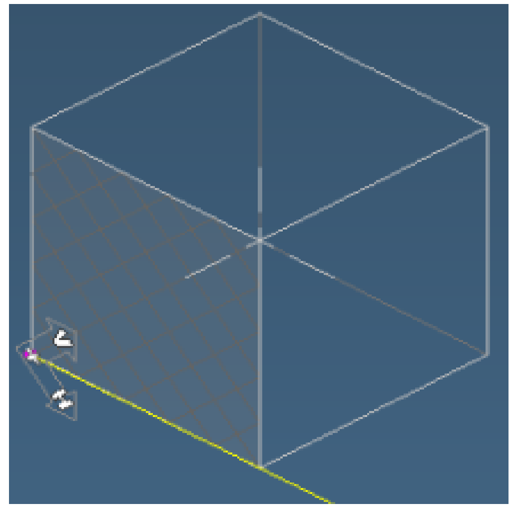
   Two additional planes are created named Learning, mod (1) and Learning, mod (2).



Single Axis Alignment Using Geometry

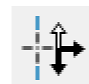
We will now use geometry to align the vertical axis of a CS.





1. Switch to CS5: Learning and hide the markers for CS4, CS6 and CS7 (`Double-click` the eyes).

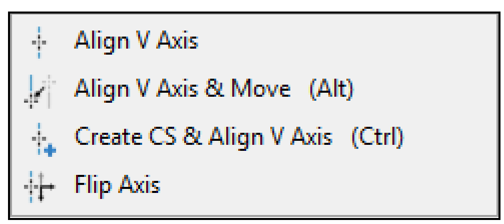
2. Create a 45° line through the point at the origin. 

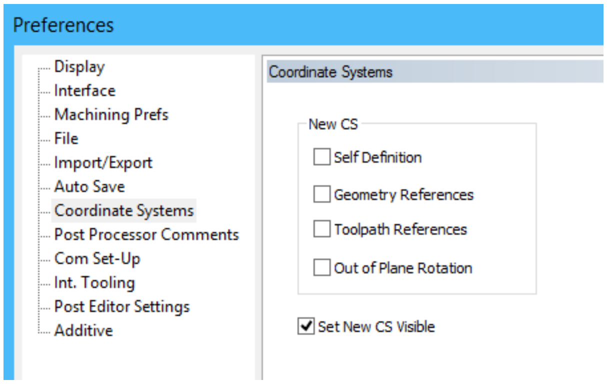
3. Select the line and `click` the Align V Axis button. 

> If you are aligning a plane to connected geometry, it is recommended that you select only the line (or two points) you wish to align the axis to. If you select multiple features (such as `Double-clicking` the geometry) the system will align the plane to the lowest numbered line.

CS5 has been aligned to the 45° line. As a result, the line has also been rotated. Geometry automatically moves with modifications to a CS. Moving the geometry is the default system behavior.

This default behavior can be avoided in several ways. The first and most common is to use an alternative aligning command by `Right-clicking` on an Align button. The default action performed when clicking on an Align button is the first item, Align V Axis. For more information on Align, refer to the Advanced CS User guide, "Align Vertical (V) Axis" chapter.



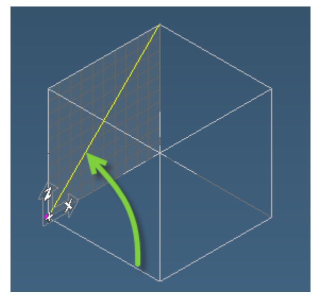


Another option is to change the default behavior of new coordinate systems. This is done through the CS Preferences dialog which is found in the File>Preferences>Coordinate Systems dialog. For detailed information on these options, refer to the Advanced CS User guide, "Coordinate System List Context Menu" chapter.

CS Alignment by Rotation Using Geometry

Rotating a coordinate system can be done by specifying a rotation about a specific axis by a given amount or can be done about geometry. We will start with geometry.

1. Select this line.



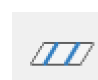
2. Click the Align CS Plane button.

Notice how the CS has flipped. A selected line is used to designate the positive direction of the depth axis. Using the Align CS Plane command with geometry will not move the origin of the CS but will rotate the CS so its depth axis is aligned to the original position of the line.

CS Alignment by Rotation Using Specific Values

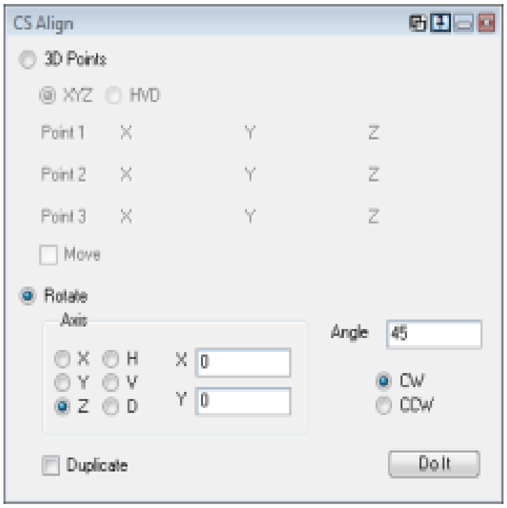
1. Deselect the line.

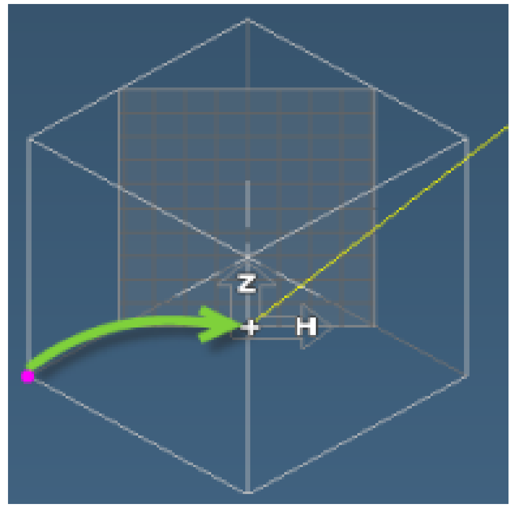
2. Click the Align CS Plane button .



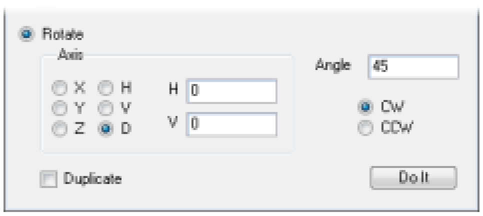
3. Enter the HVD values shown and Do It.

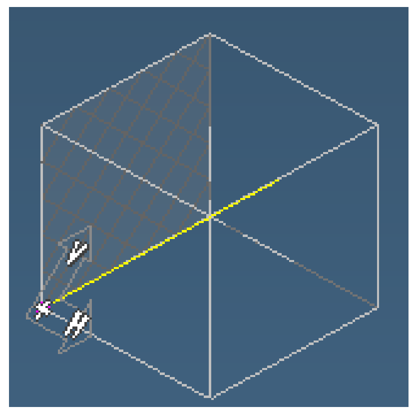
We have specified that the coordinate system is going to be rotated about X0Y0 by 45 degrees in a clockwise direction using absolute (XYZ) values.





If we had specified the rotation was to use relative (HVD) values, the dialog would have displayed H0 and V0 and the results would look like the image shown.

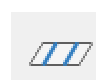




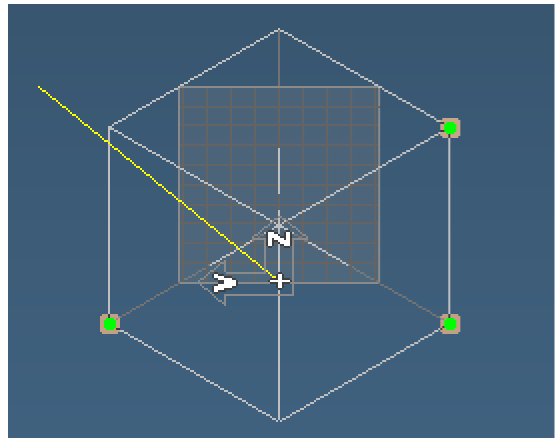
CS Alignment Using Geometry

Geometry used to align a plane may be a set of points, a shape (open or closed), or a combination of lines and points. Here we will first use a set of points to define the plane's alignment, then we will use a closed shape. Please note that if the transformations shown here do not work then you may need to change your CS Preferences.

1. Switch to CS2: XZ plane.
2. Create points at X100, Z0 and X100, Z100.
3. Select the two points as well as the point that lies in the YZ plane.
4. Switch to CS5: Learning and `click` the Align CS Plane button .

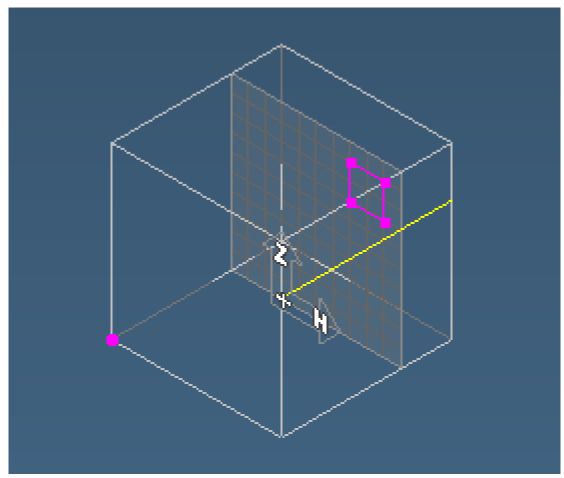


The CS remains essentially unchanged, it has only rotated 90°. That is because the plane is already parallel to the selected points. Any additional clicks on the Align CS Plane button will continue to rotate the CS by 90° increments.



5. Undo (`Ctrl+Z`) the align and Delete the two points we just created.
6. In the YZ plane, create a 20mm square around Y50, Z50.
7. Switch to CS5: Learning, `Double-click` the square and `click` the Align CS Plane button .

The origin of the plane stays in the same position but the plane swings around to be parallel to the geometry.

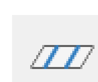


> ⓘ To move the origin when aligning a CS, one of the move options must be chosen from the Align CS Plane context menu (right-click).

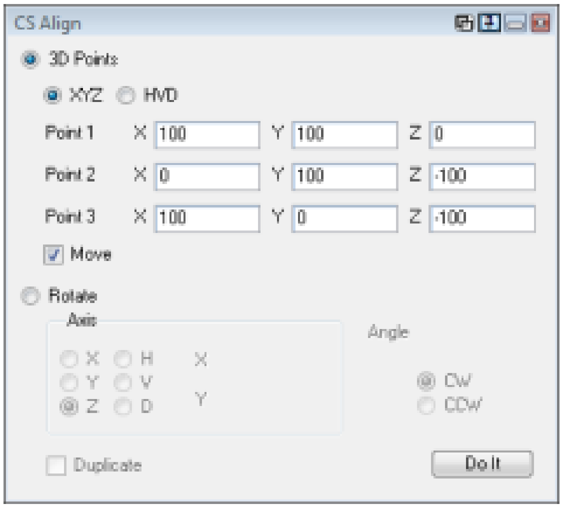
CS Alignment Using 3D Points

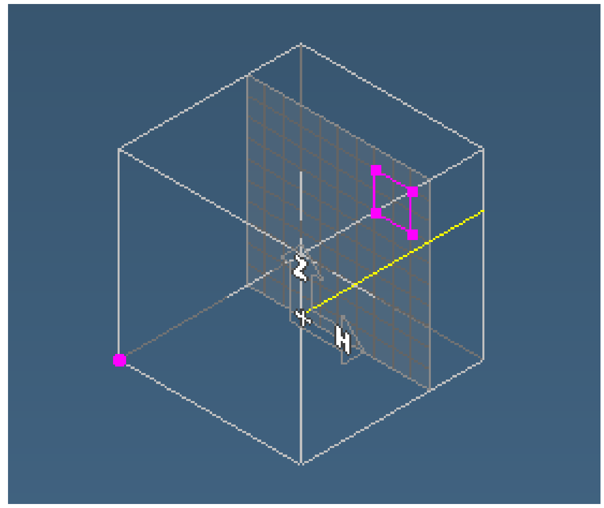
The last CS Alignment exercise will be user-defined 3D points.

1. Deselect all geometry and `click` the Align CS Plane button .



2. Enter the values shown, making sure to use the XYZ values and the Move option before `clicking` on Do It.

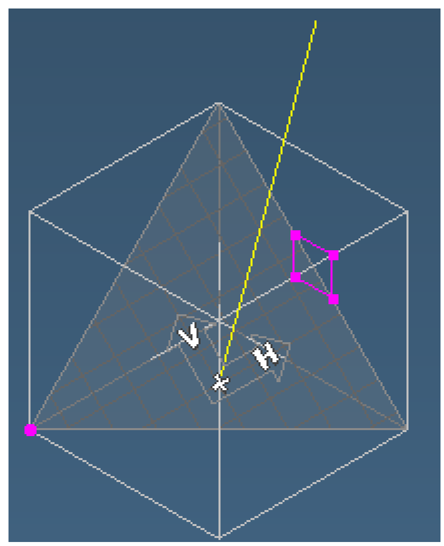




The results are an angled plane. the plane aligns to the top-right corner of the XY plane (X100,Y100,Z0), and the top-left and bottom-right corners (X0, Y100, Z-100 and X100, Y0, Z-100) of the bottom of the part.

ⓘ Existing points can be interrogated ((`Alt+click`) and (`Shift+Alt+click`)) to fill in data but the retrieved data is relative to the current coordinate system. If you want absolute data be sure to switch to the XY plane, interrogate the points and then switch back to the coordinate system you are going to modify.

Because we selected the Move option, the depth origin of plane was moved to align with those points. If we had not selected Move, the results of the CS alignment would look like this image. The plane does align with the points, but the origin remains where it was.



You may now save this part if you wish, but it will not be used in any other tutorials.

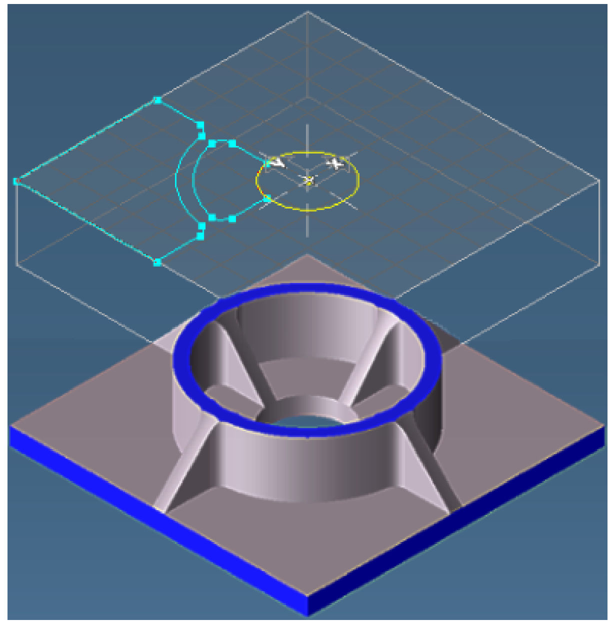
3-Axis Part: Angled Webs

In this exercise, we will create a part that has angled webs that require following a 3D contour in order to machine it properly. The Advanced CS option is necessary to generate this type of toolpath because the contour that we will be machining lies in a plane other than the standard XY plane. Creating the contour in another coordinate system allows the system to be able to correctly offset the tool cutting in Z. We will machine the part from the XY plane. This type of operation can be created on any contour that lies in a plane that has Z as an axis. Providing that Z is an axis in the plane, the system can follow a 3D contour, correctly offsetting the tool. For the Part Print for this exercise, see “Angled Webs” on page 105.

Creating the Angled Webs

About the Part File

There is a pre-existing part file for this exercise. The file name is Angled webs.vnc and it may be found in the install directory for the part files or on your GibbsCAM installation CD. If for some reason you do not have the part you may re-create it using the part print.



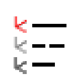
Creating the XZ Plane

1. Open the file Angled Webs.vnc.

   The part is 200 x 200 x 50mm set up on a 3-Axis Vertical Mill MDD. It contains three tools – a 45mm drill, a 13mm Rough endmill and a 45mm rough endmill. There is geometry representing one set of the open pockets and the thru-hole. There are also pre-generated operations to clear out the hole and the open pockets. As you can see, there is only one set of geometry for the open pockets. The Transform Ops plug-in was used to duplicate and rotate the toolpath.

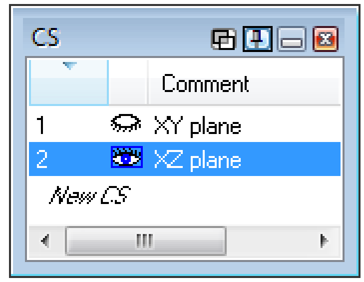
   We need to create the contours that will generate the angled webs. In order to do this, we need to make two additional coordinate systems, the XZ plane and the YZ plane. The web contours will be drawn in these two planes.

2. `Click` New CS in the CS list.



3. Modify the CS to the XZ plane and rename it `XZ plane`.

The polarity of the depth axis does not need to be adjusted because we will be machining these shapes in the XY plane.



Creating the YZ Plane

Two of the web contours need to be defined in the YZ plane to provide for proper machining.

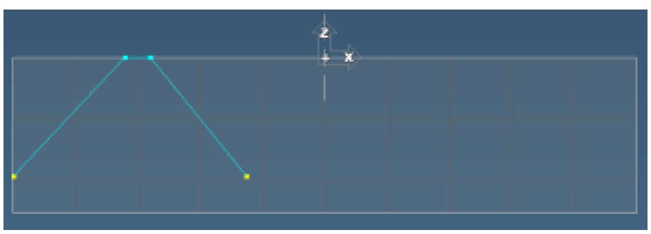
1. Create a new CS by `clicking` New CS in the CS list.

2. Modify the CS to the YZ plane and name it `YZ plane`.

Creating the Geometry

To create the part geometry we will create one profile of the angled web. We will then Duplicate and 2D Rotate the geometry about the origin.

1. Switch to CS2 XZ plane. Create a new workgroup and create the contour geometry.

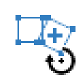


From the home view (`Ctrl+H`) of the XZ plane, your screen will look like the screenshot on the left. The home view always shows the part along the positive depth axis of the current CS.

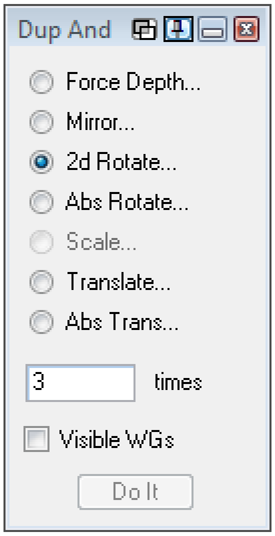
The web contour needs to be duplicated in the four locations that need to be cut.

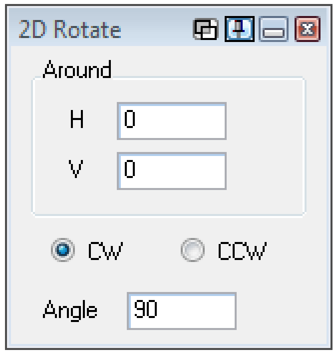
2. Switch to CS1 and `Double-click` the web contour geometry.

3. Select Modify > Duplicate And 2D Rotate

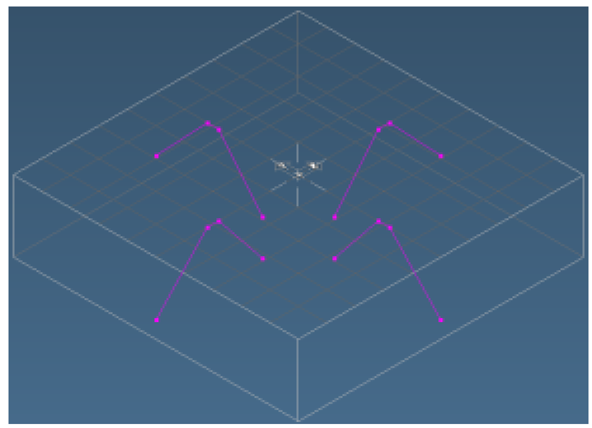


4. Enter the following.





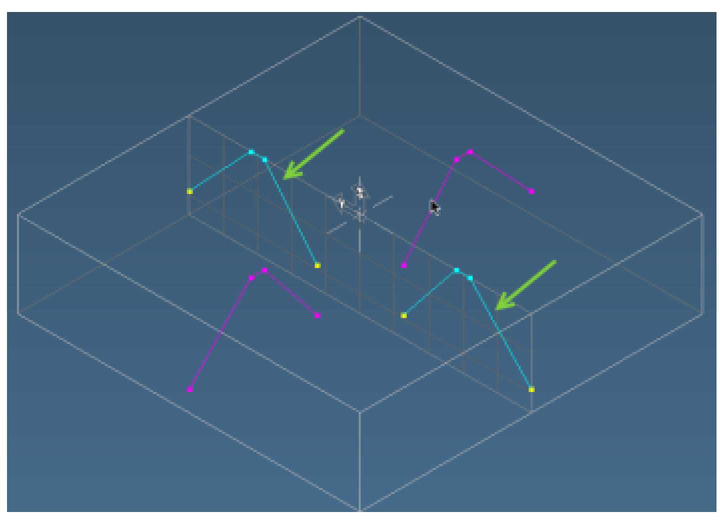
We have duplicated and rotated the geometry at a 90° angle, 3 times.



Aligning Geometry to the YZ Plane

1. Switch to the YZ plane.

2. Select the two web contours shown.



3. Select Modify > Change CS (XYZ).

By using Change CS (XYZ), the shapes will stay in the same location but will now be drawn in cyan, rather than magenta, to indicate that they are defined in the current CS.

The geometry and coordinate system creation is complete. For more information on creating the machining operations, see “3-Axis Part: Angled Webs” on page 38.

4-Axis Part: Slot Block

In this exercise we will create three new coordinate systems. The first CS will be one of the primary planes, the YZ plane, and the others will be created by rotation using the Align CS dialog. For the Slot Block Tutorial exercise, see “Slot Block” on page 106.

Creating the Slot block

Part Setup

1. Create a new part with the following stock dimensions:
   +X = `125`, –X = `-125`, +Y = `250`, –Y = `-250`, +Z = `0`, –Z = `-150`.

   This part will be machined later in the Machining tutorials. At this point in the tutorial it does not matter if a specific MDD is used but for consistency, we shall choose one.

2. Select a 3-Axis Vertical Mill MDD.

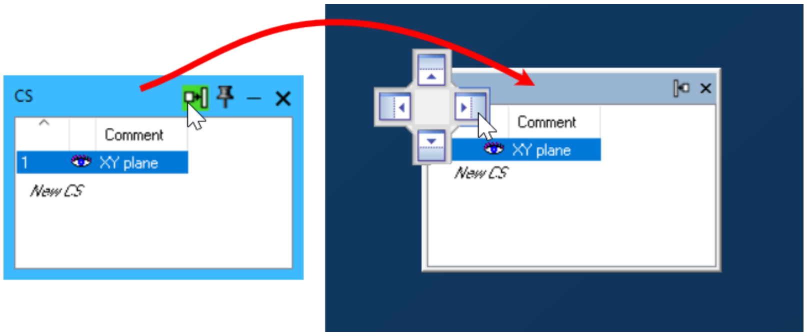
Back Profile

1. Switch to the isometric view (`Ctrl+I`).

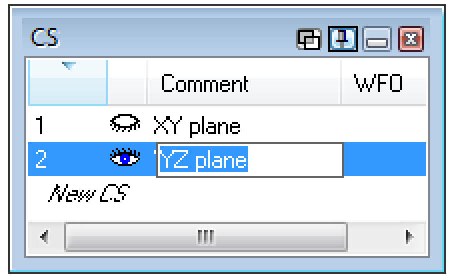
2. Open the CS Dialog

3. Attach the CS dialog to the right side of your screen by clicking the docking icon on the dialog. Now click and drag the title bar, placing the cursor arrow on the grid selection as shown below. The CS dialog snaps to the outside of the Gibbs screen, keeping the workspace clear.

   We will be using the CS dialog extensively during this tutorial.

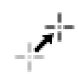


.

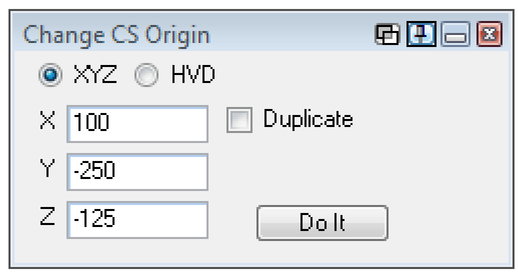


4. Create a New CS and name it `YZ Plane`.

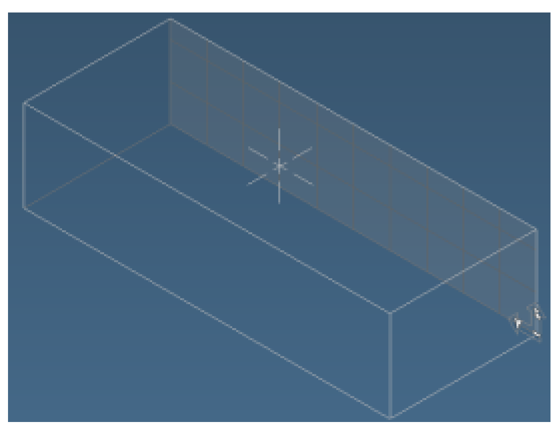
5. `Click` the YZ button in the CS palette.



6. `Click` the Change CS Origin button.



7. Enter the information shown and Do it.



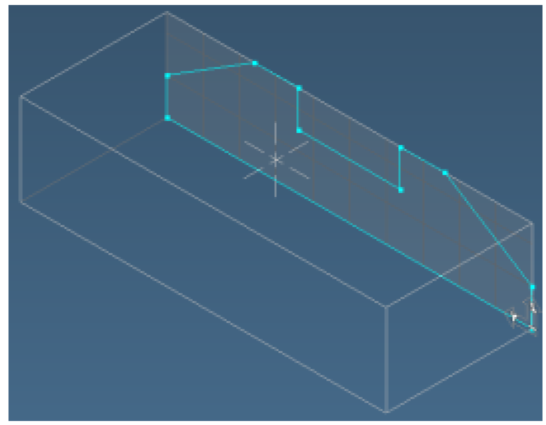
8. Switch to the home view (`Ctrl+H`).

When working in the home view, you are viewing the part from a position normal (perpendicular) to the current coordinate system. Geometry creation can proceed from this point as if you were working in 2D. The only difference is the labels for the horizontal, vertical and depth axes (coordinates). In this case, the horizontal axis is labeled as the Y axis, the vertical axis as the Z axis, and the depth axis as the D axis (not the Y axis because the depth axis polarity is reversed).

> Not only is the orientation of the depth axis important for the machining of the part, but also many of the solid modeling functions in the system are dependent on the depth axis, such as extrusions and revolutions.

9. Create the geometry for the YZ Plane.

Because we set the origin in the position that we did, you should be able to use the dimensions provided on the blueprint. Remember the origin acts as a zero reference mark for dimensioning. Make sure that the D (depth) value in the geometry dialogs is `0`.

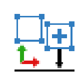


10. Switch to the isometric view (`Ctrl+I`).

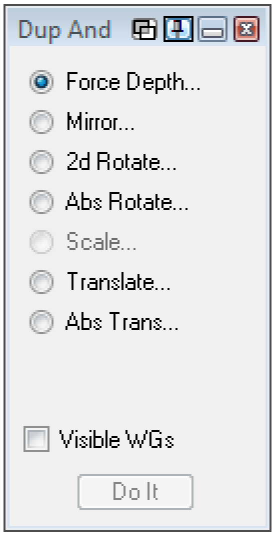
Front Profile

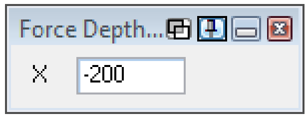
Now, we will duplicate the shape along the depth axis.

1. Select the entire closed shape created in the YZ Plane by `Double-clicking` on it.
2. Select Modify > Duplicate And....

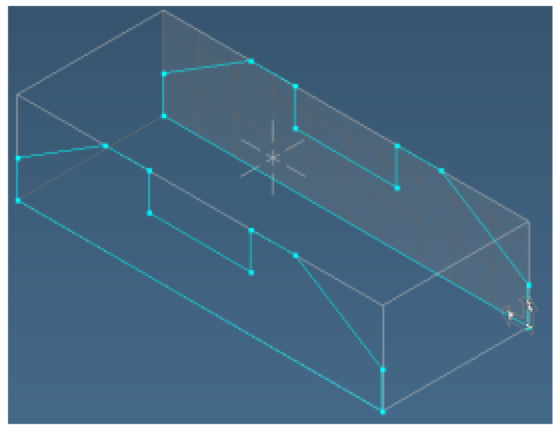


3. Enter the value shown.





This will duplicate the selected geometry and move it the specified amount along the depth axis. We could also have used the Translate item in the Dup And dialog to accomplish the same thing.



Right Side Angled Face

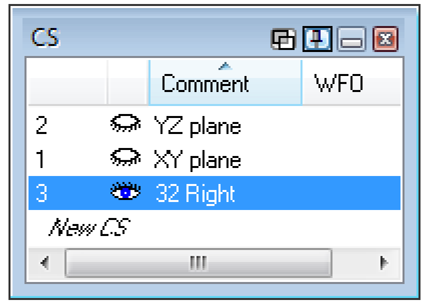
Now, we will create the 32° plane that lies on the right side of the part. There are two primary methods for changing the orientation of a CS to a non-standard plane. You can either select plane group geometry, which is the minimum amount of geometry necessary to accurately define a plane, or you can use the Align CS dialog, which allows you to enter specifications to properly orient the plane. We will use the Align CS dialog to rotate the plane to the correct orientation.

1. Switch to CS1.

   We have switched to CS1 so that the new CS will be a duplicate of the XY plane rather than the XZ plane.

2. Create a New CS and label it `32 Right`.

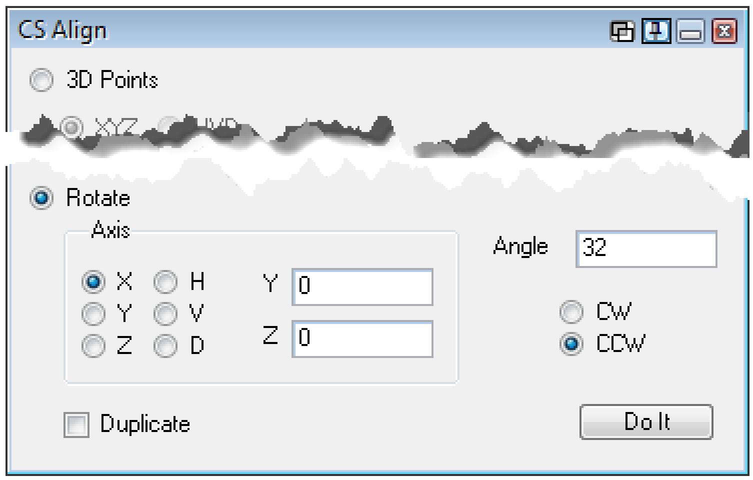
Make sure all geometry is deselected before clicking on the Align CS button.



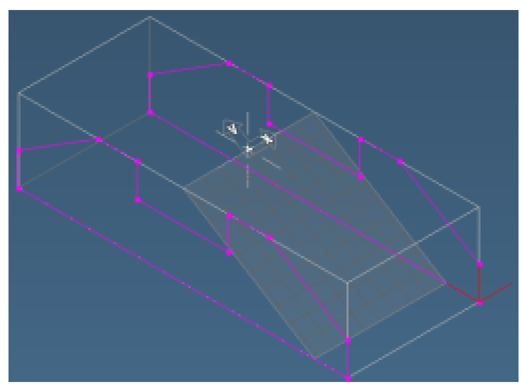
3. Click the Align CS button in the CS palette.

The Align CS dialog will appear on the screen. If it does not, you have geometry selected and the system is attempting to orient the plane according to the selected geometry.

4. Select Rotate and enter the information as shown.



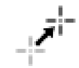
The system will modify the new coordinate system by rotating it from its current orientation, which is the standard XY plane. In this case, the CS will be rotated 32° about the X axis in a counterclockwise direction. The location of the X axis which will be rotated about is determined by the values entered in the Y and Z text boxes.

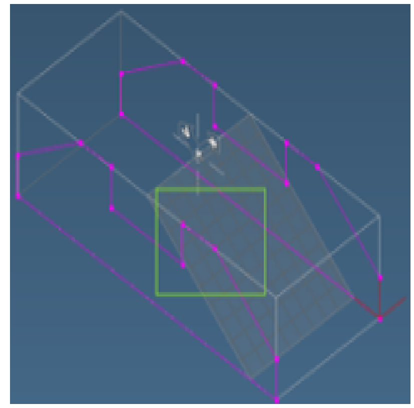


The planar orientation of the coordinate system will change, but the origin will not be affected. To further modify the new CS to properly create the slot geometry, we will need to change the origin.

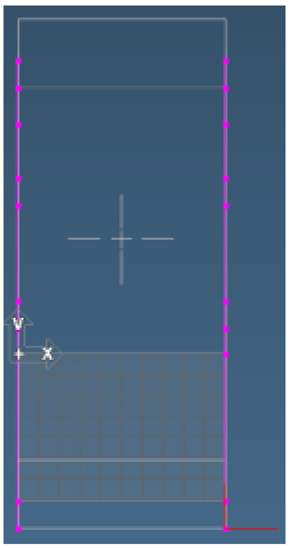
We can change the origin of a CS by entering coordinates in the Change CS Origin dialog, which is how we modified the last CS. Another method for changing the origin is to select an existing point and click the Change CS Origin button. This action will cause the selected point to become the new origin for the CS. In this case, we will select an existing point to change the origin.

5. Select the point shown and `click` the Change CS Origin button .

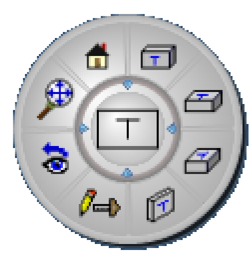




6. Switch to the home view (`Ctrl+H`).



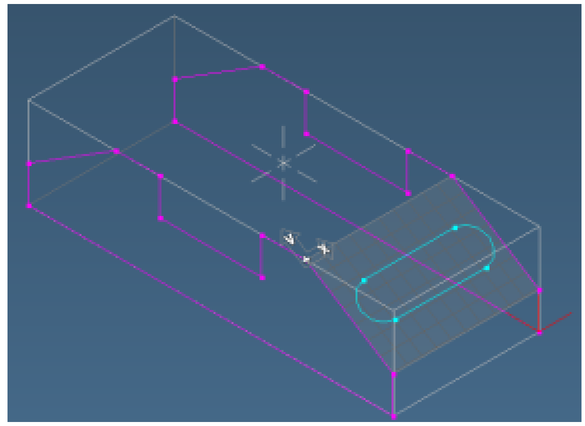
In order to get an idea of the orientation of the part, look at the trackball in the center of the View Control palette.



This gives you an indication of the view at which you are looking at the part and also the orientation the part will be in when it is machined.

7. Create the slot geometry.

The labels in all of the geometry dialogs should read H for the Horizontal axis, V for the Vertical axis and D for the Depth axis as none of the axes in the current coordinate system align with the primary X, Y and Z axes. Make sure the CS grid and axis markers are drawn on the screen so you can keep in mind how the part dimensions should be entered.



8. Switch to the isometric view (`Ctrl+I`).

Left Side Angled Face

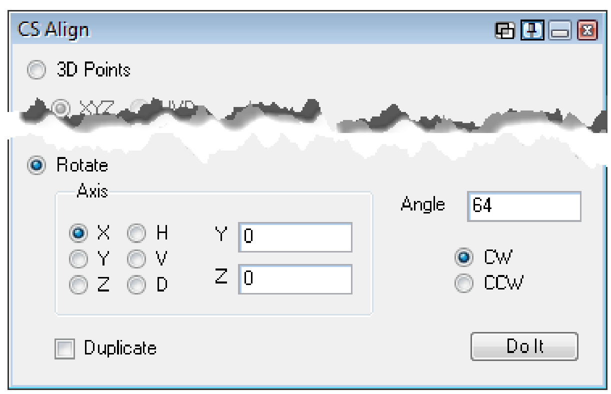
Now, we will create the 32° plane on the left side of the part.

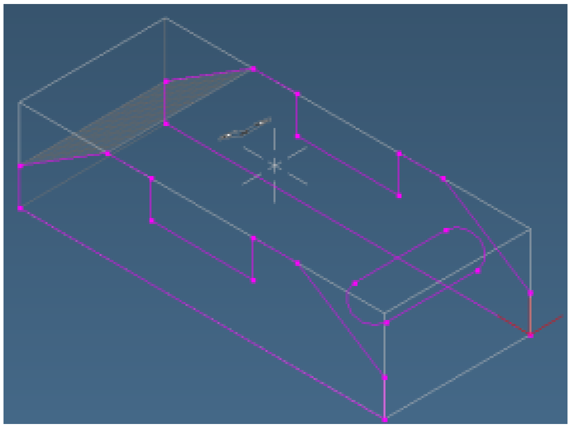
Make sure that the current CS is the 32 Right plane so that the new CS is a duplicate of that plane.

1. Create a New CS and name it `32 Left`.

2. Open the Align CS dialog.

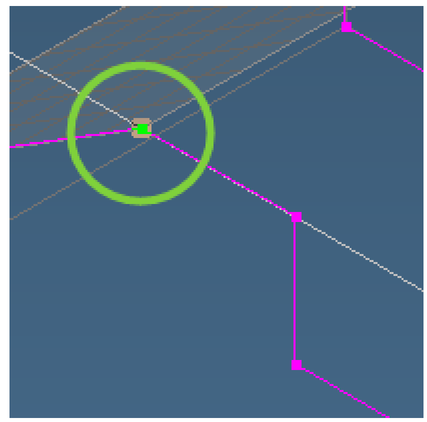
3. Enter the information shown.





We have rotated the current CS a total of 64°. A 32° rotation would have oriented the plane so that it were parallel to the XY plane. The additional 32° places it in the proper orientation for the left side of the part.

4. Select the point shown and change the CS origin. 



It may be difficult to see the labels on the axis markers in the current view and specifically to tell the polarity of the depth axis (indicated by the "+" or "-" sign on the axis markers).

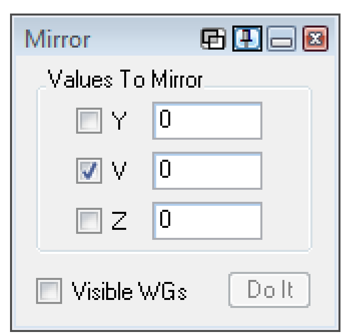
5. Ensure that you are in CS4, the 32 Left plane.

6. Select the right hand slot geometry and duplicate it.

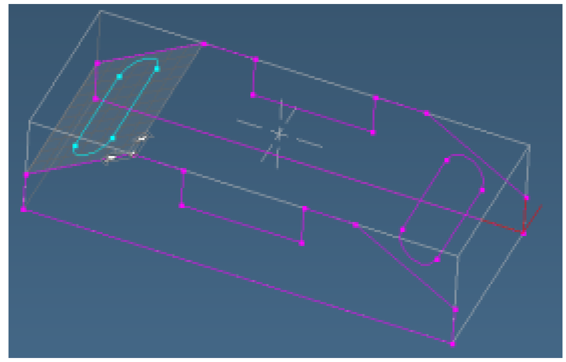
7. Select Modify > Change CS (HVD).

This moves the geometry around to the new CS and the arcs are not converted to splines, which can happen when performing certain modifications to geometry.

8. Mirror the slot geometry about V0.



The final part geometry will look like this.



5-Axis Part: Hinge

In this exercise, we will create four coordinate systems, using a normal vector to define one of the planes. For the Hinge part print for this exercise, see “Hinge” on page 107.

Hinge Set Up

Part Dimensions and Machine Type

- Create a new part `Hinge.vnc` with the following stock dimensions: +X = 150, –X = 0, +Y = 75, –Y = 0, +Z = 0, –Z = -90.
- Select a 5 Axis Horizontal Mill for the Machine.

  We will not go into the rotary setup for the part in this section. For a more detailed description of rotary setup, see “Rotary Setup” on page 48.

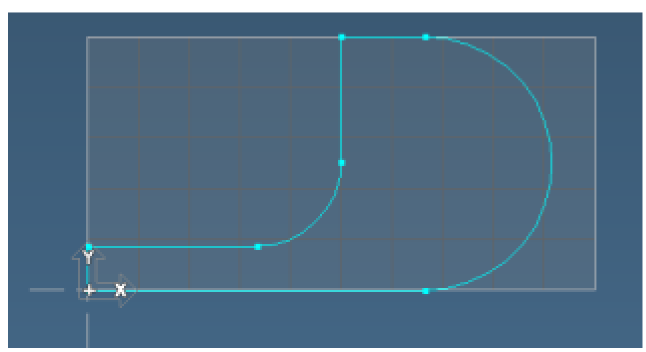
Creating the Hinge

Top and Bottom Geometry

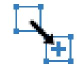
The first thing we will create is the part’s top profile. We will then duplicate the geometry and change its depth to create the bottom profile.

1. Create the top view geometry in the XY plane.

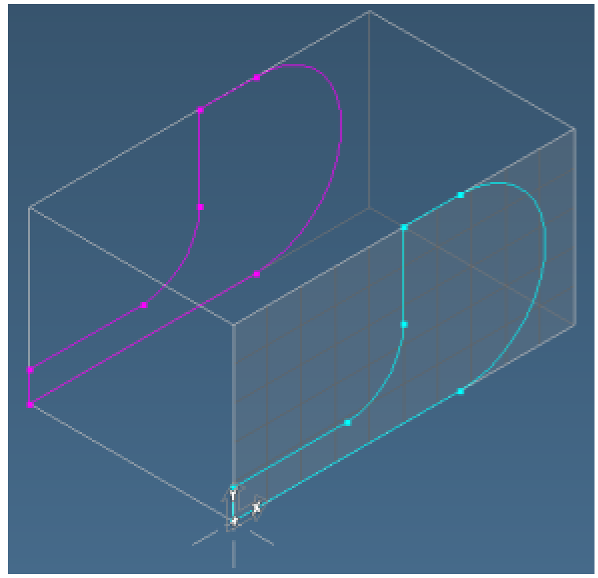
   The top view geometry we want to create is the rounded contour.



Now, we will duplicate this shape to create the back side geometry for the hinge.

2. Select the entire contour then Dup And Translate the shape to a depth of Z-90. 

3. Switch to the isometric view (`Ctrl+I`).



Back Side CS Creation

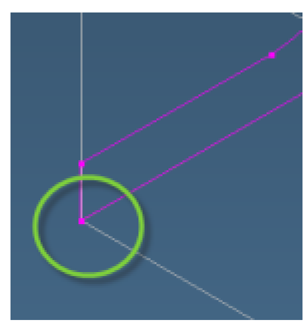
We will now create a new CS in preparation for the machining of this part. When creating machining operations, the tool approaches and cuts the part along the positive depth axis of the CS selected as the Machining CS. The standard XY plane does not have the depth axis in the correct position to machine the back side of the part.

1. Create a new CS and name it `XY back side`.

This will be an exact duplicate of the XY plane, which is the correct orientation. We simply need to change the origin and toggle the depth axis.

Assigning the Back Side Geometry

1. Select the point shown and `click` the Change CS Origin  button.



2. `Click` the Toggle Depth button . 

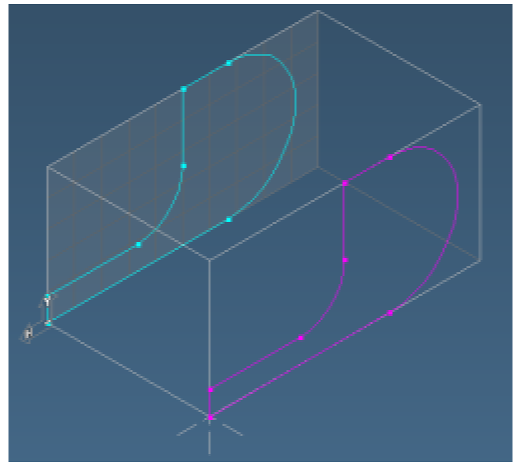
`Clicking` the Toggle Depth button will flip the depth axis which also has the effect of inverting the horizontal axis. Any rotations of the coordinate system axes, follow the Right Hand Rule.

Notice that from this view, you see a "-" sign on the axis markers indicating that you are viewing the part along the negative direction of the depth axis. Use the home view and the trackball to get an idea of the orientation of the CS and the part.

You will notice that both shapes are magenta. This indicates that these shapes were created in a different coordinate system.

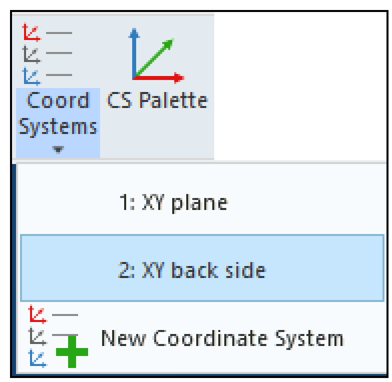
3. Select the entire shape that was just moved and select Modify > Change CS (XYZ).

This shape should now be drawn in blue on the screen, indicating that it was created based on the current coordinate system. The Change CS (XYZ) function assigns all selected geometry to the current coordinate system. The geometry stays in the same location in 3D space in which it was created.



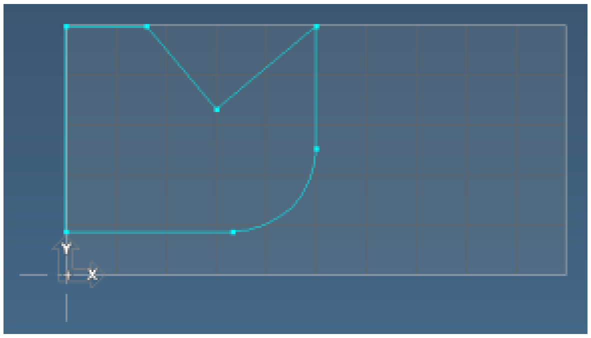
Center Wall Creation

We will now create a new workgroup the geometry for the angled center wall. This geometry will be created in the standard XY Plane, CS1. You can display the coordinate system list by `clicking` the Coordinate system list down arrow button.



1. Create a new workgroup and switch to the XY Plane.

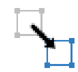
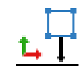
2. Create the center wall geometry.



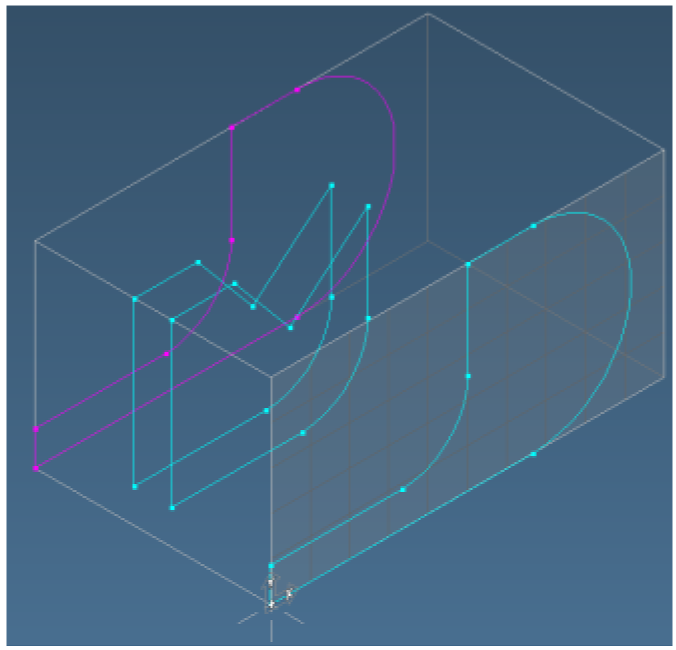
You do not need to worry about creating the geometry at the appropriate depth because we can use Modify > Force Depth or Modify > Translate.

Note that it is not necessary to draw the entire shape that is shown in order to properly program this part. However, we are drawing this for visualization purposes.

3. Use Modify > Translate or Force Depth to place this shape at a depth of Z: `-38` and Z: `-52`mm.

4. Cut and paste this geometry into WG1.

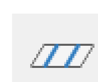


Pocket Geometry CS

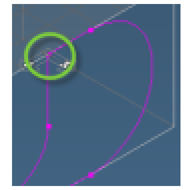
We will now create geometry for a pocket in the XZ plane. The pocket will create the separation of the two sides of the hinge. In order to make the geometry we need a new coordinate system.

1. Create a new CS named `XZ top side`.

2. Align the new CS to the XZ plane.



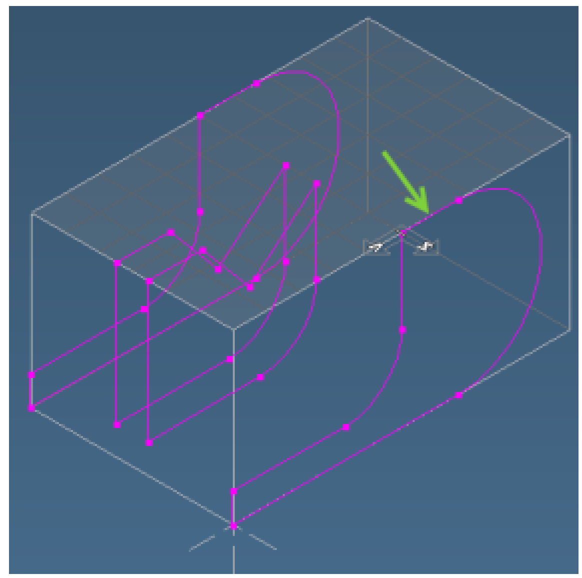
We will modify this new CS to be flush with the top of the part, with its vertical axis aligned to the front profile geometry.

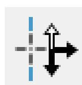


3. Switch to WG1 and align the CS origin  to the point shown.

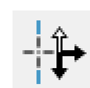
4. Toggle Depth axis . 

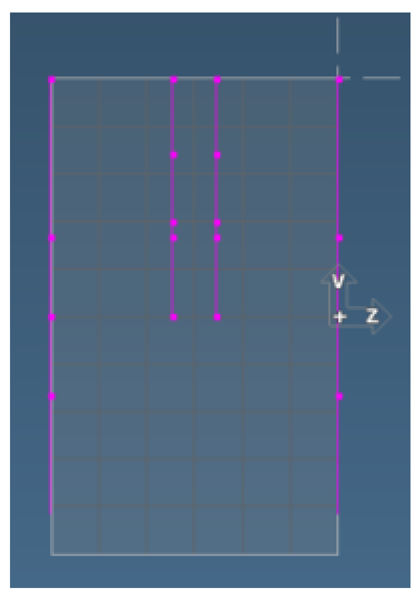
   The CS should now be aligned to the Y+ side of the part. To correctly orient the axes of this coordinate system, we will use the Align Vertical Axis button.



5. Align Vertical Axis  to the line shown.

   This will rotate the axis markers so that the vertical axis aligns with the selected line.

6. Make sure the line is still selected and `click` the Align Vertical Axis button again. 



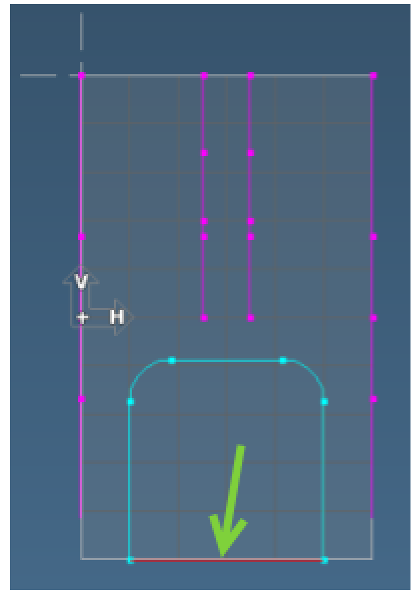
7. Switch to the home view (`Ctrl+H`).

Our plane is now correctly oriented.

> Once an axis has been aligned, additional `clicks` on the Align Axes buttons with geometry selected will flip the alternate axis 180°.

Pocket Geometry Creation

1. Create the pocket geometry as shown.



The pocket geometry should be flush with the end of the part. The line that is on the edge of the part should be designated as “Air” geometry so that when we machine the geometry, the tool will overlap the geometry, creating an open pocket.

At this point, you will have three coordinate systems and two workgroups. Separating geometry into different workgroups is not necessary; however, it often simplifies geometry creation.

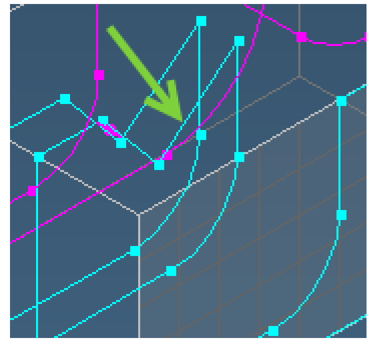
Angled Hole CS Creation

To complete the part, we need to create one more CS for the hole on the angled wall.

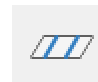
1. Create a new CS based on CS3 named `Hole`.

We will use a normal vector to correctly orient this plane. Normal vectors are perpendicular to the plane.

2. Select the line shown which is the normal vector to the CS we wish to create.



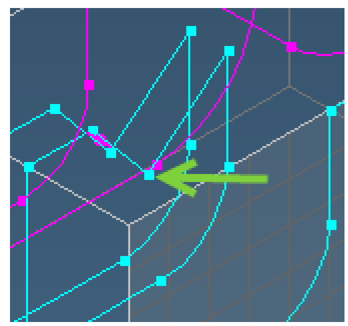
3. `Click` the Align CS button.



The Align CS button will attempt to align the plane through the selected geometry. In the case where not enough geometry is selected to accurately orient the plane, the system will attempt to align the plane normal to the selected geometry which is what we have done in this case. The modified CS will be perpendicular to the line we selected.

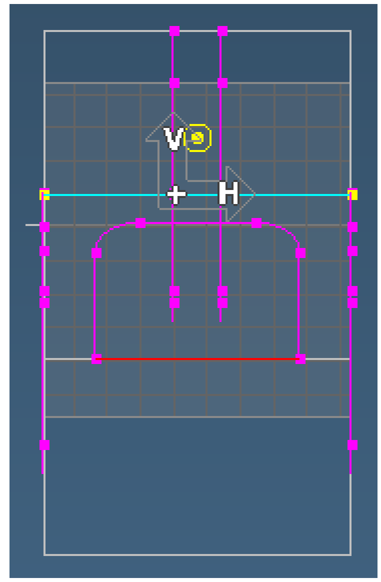
4. Select the point shown.

The point is the vertex of the 90° angle on the center wall.



5. `Click` the Change CS Origin button .

The screenshot on the right shows the correct placement and orientation of the plane in the top view. The plane should be correctly oriented with the positive depth axis pointing out indicated by a "+" sign at the center of the axis markers. If it is not, toggle the depth.



6. Switch to the home view (`Ctrl+H`).
7. Create the circle for the drill hole.

   The geometry for this part is now complete.

8. Save this file (Hinge.vnc) as it will be machined in the Machining section.

MACHINING CS'S

This chapter describes the uses of the Advanced CS option with a 3-axis, 4-axis, and 5-axis machine. The parts described in this chapter require the creation of multiple coordinate systems to properly define the part geometry and provide for accurate posted output. Multiple part programming and tombstone programming will also be covered.

This chapter is divided into two sections. Parts in both sections use multiple coordinate systems for machining and do not involve rotary positioning.

In the first section, all of the parts are created using a 3-axis Machine Type and are posted with a Standard Mill Post Processor. Coordinate systems are used to define part geometry in the appropriate locations. Once that is done, the parts are programmed with the system as they would be, using the Production Mill Module.

The second section uses coordinate systems to utilize work fixture offsets for machining multiple parts and/or multiple sides of parts in one program. These examples require an Advanced CS Post Processor to produce the appropriate posted output which uses G55, G56, etc. for the work fixture offsets. These exercises can be programmed to run on a 3-axis milling machine and use the standard XY plane for all of the machining operations. However, an Advanced CS Post Processor is required to use work fixture offsets.

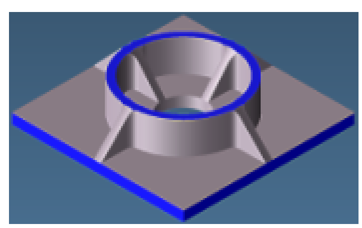
- 3-Axis Part: Angled Webs
- 4-Axis Part: Slot Block
- 5-Axis Part: Hinge

3-Axis Part: Angled Webs

This exercise illustrates that contours that are defined in any Z plane (a plane where Z is an axis) can be machined from the XY plane. This part will be machined with a 3-axis mill.

About the Part

For the detailed description of the geometry and coordinate system creation for this part, see "3-Axis Part: Angled Webs" on page 18. The part is created using a 3-Axis Vertical Mill MDD. Some of the machining operations for this part are pre-generated and all of the required tools are already in the tool list. This includes a 45mm Drill, a 13mm Rough Endmill and a 45mm Rough Endmill.

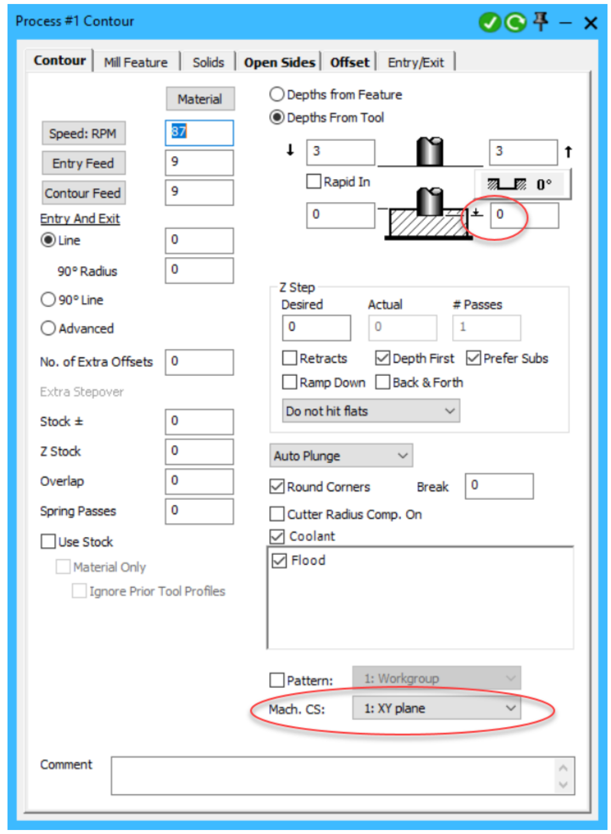


Machining the Angled Webs

The First Contour Process

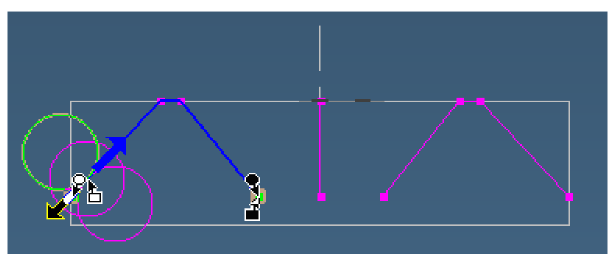
The only area that remains to be cut are the four webs to create the angled walls. Thus far all of the machining operations could have been done with a standard mill. The operations we need to create to cut the webs will follow the contours we created in the XZ and YZ planes.

1. Open the file Angled Webs and create this Contour process with tool #3.



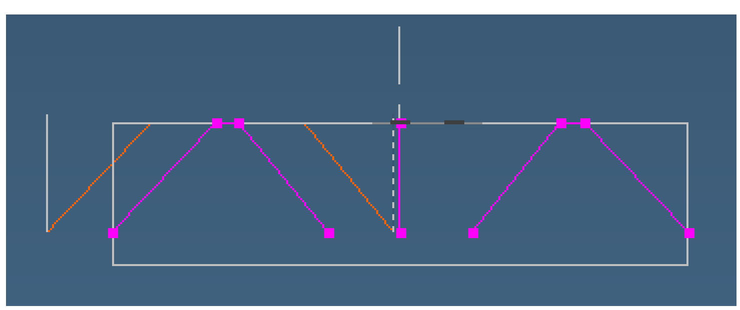
Notice that the Floor Z value is 0. This way the toolpath will follow the contour and the depth of cut will be determined by the selected contour. The Floor Z value in the process dialog controls the depth value in reference to the geometry's coordinate system. The value of 0 is good to use if the geometry is at a depth of 0 in the CS and the CS is in the correct position. Also, the Machining CS is the XY plane, not the plane that the contour was defined in. All of the web contours will be machined from the XY plane.

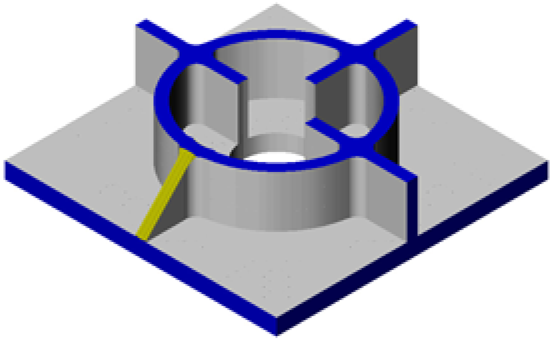
2. Select the web contour from CS1 as shown.



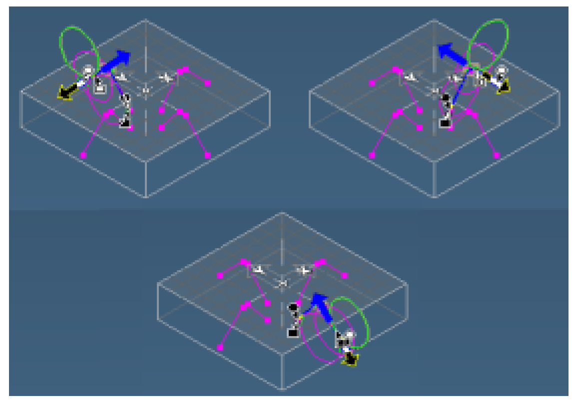
3. `Click` the Do It button or use (`Ctrl+.`).

   The screenshot is displayed from the Front view. The current CS is the XY plane.



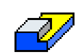


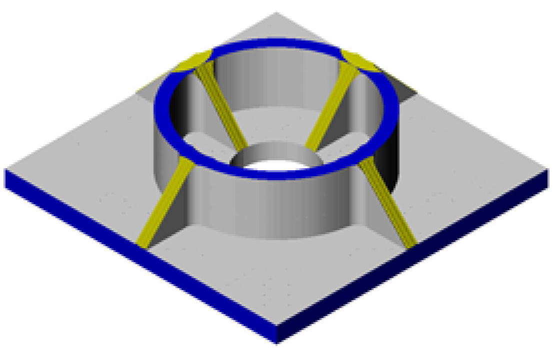
4. Deselect the operation and repeat the process for the other three webs as shown.



The information in the Contour process data remains exactly the same. Simply select the cut shapes and properly position the machining markers.

5. Render the contour operations.



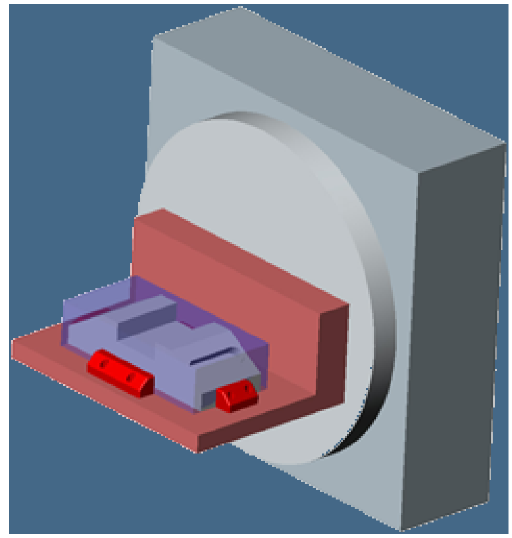


The part is now complete.

4-Axis Part: Slot Block

Defining the Slot Block

For a recap of the coordinate system construction and geometry creation for this exercise, see “4-Axis Part: Slot Block” on page 22. In this exercise we will pocket the center of the part and we will face mill and pocket both of the 32° coordinate systems. For the Slot Block part print for this exercise, see “Slot Block” on page 106.

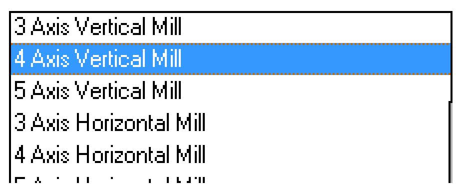


Setting Up the Slot Block

In order to face mill and pocket the two 32° planes we need a machine with a rotary axis. We will machine this part on a 4 Axis Vertical Mill. While there are several ways this could be set up, we will create the part as if it were mounted to the machine on a fixture plate. This is illustrated in the above image. The part is shown inside a transparent stock body which is mounted to the fixture plate.

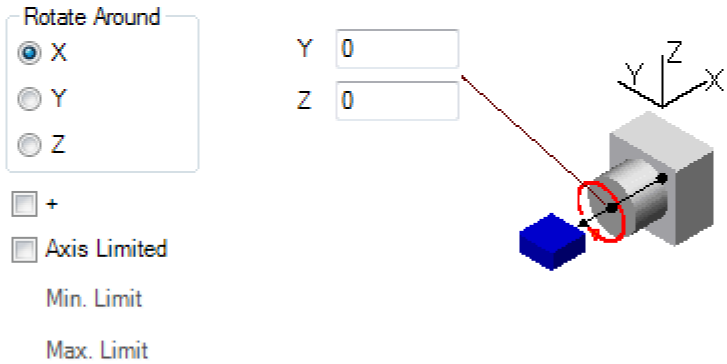
1. Open the Slot Block.vnc file.

2. Open the Document Control dialog and select the 4 Axis Vertical Mill MDD.



3. Adjust the stock values to X+100, -100, Y+250, -250, Z+ 0, -125. Set the Clearance Plane to 25mm.

4. Click the Machine Setup tab and enter the information shown.

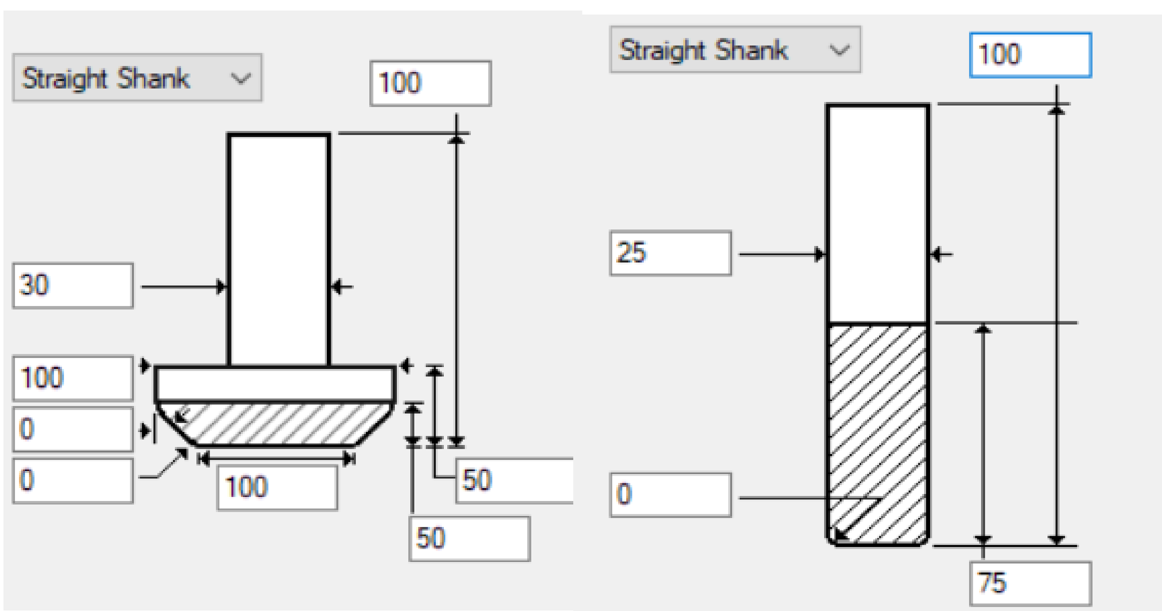


This specifies that the point of rotation is the part origin. This would require that the part be carefully positioned on the fixture plate such that Z0 is actually at the center of rotation.

Machining The Slot Block

Tool Definition

We will use only two tools to machine this part, a Face Mill and a Rough Endmill.



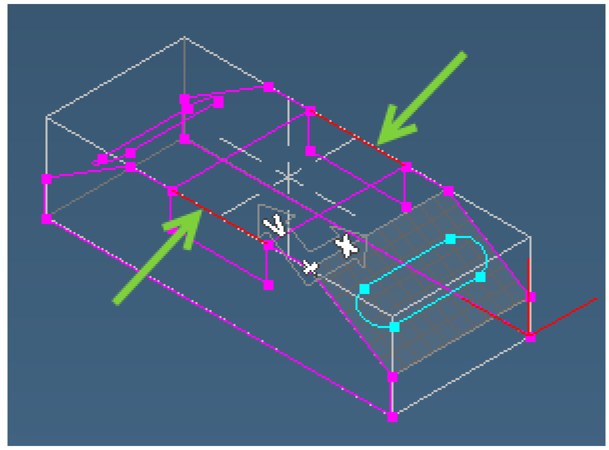
1. Create a Face mill with a 100mm diameter and a 50mm flute length. This is a flat bottomed tool.
2. Create a rEM with a 25mm diameter. This is also a flat bottomed tool.

Center Pocket Geometry

The first operation we will create will be a pocketing operation to rough out the center portion of the block. In order to do this we will need to create the pocket geometry.

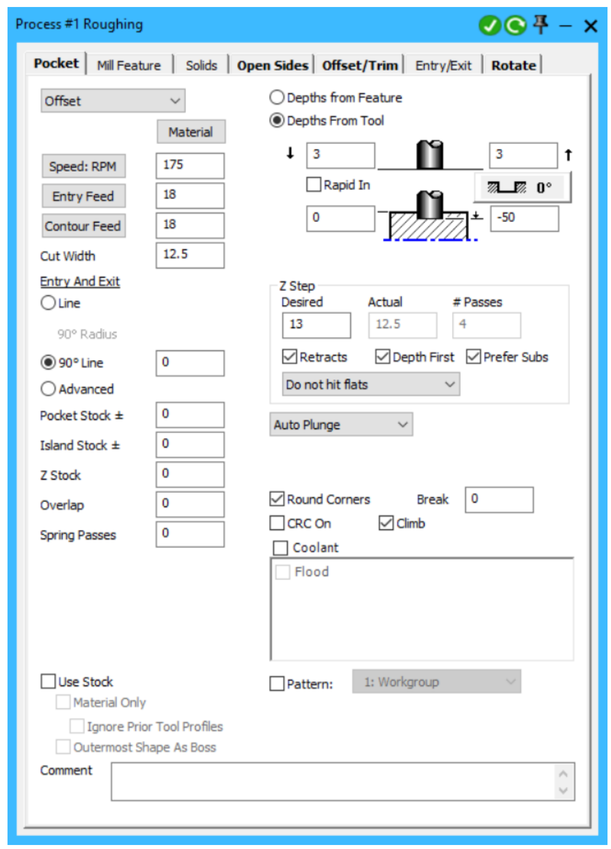
1. Create 200x140mm rectangle at the origin in the XY plane. Define the left and right sides of the rectangle as "Air" geometry so that we can easily machine this open pocket.

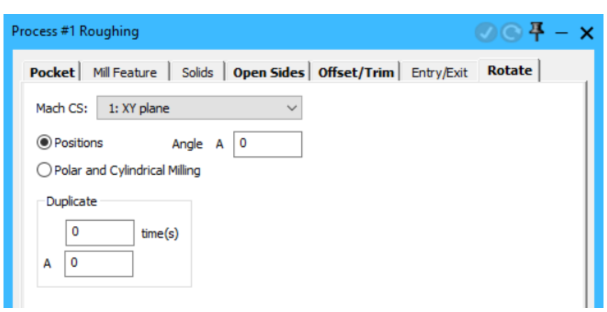
Notice the smaller sides of the rectangle are designated as "Air" geometry so that the pocketing routine will remove all of the necessary material.

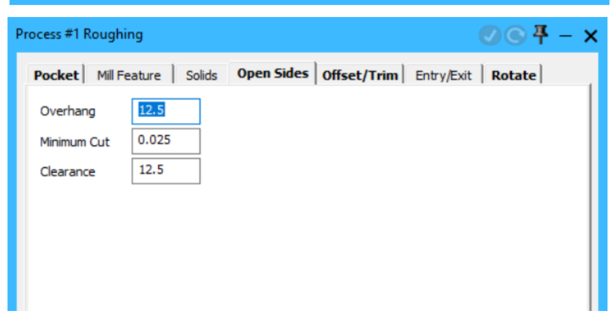


Operation 1, Pocketing the Middle

1. Create this Roughing process with the Rough Endmill.

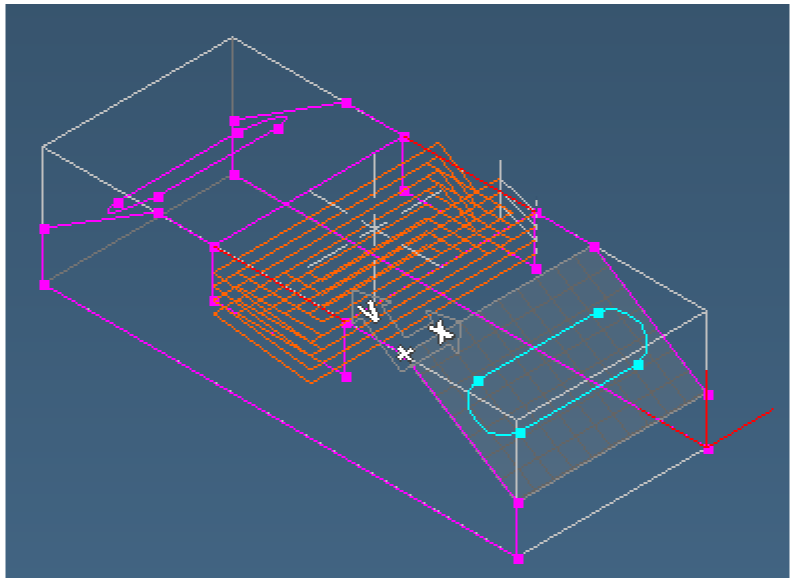






2. In the Rotate tab ensure that the Mach CS is set to the XY plane. The Open Sides data does not need to be changed, the default values are adequate.

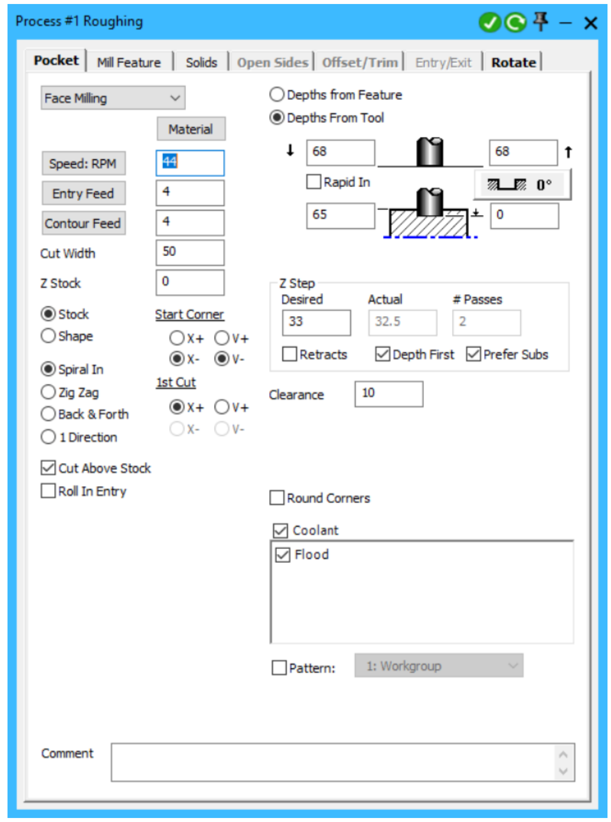
3. Select the pocket geometry and create the operation.



Operations 2 and 3, 32° Right Face

We will now create operations to face mill and pocket the 32° planes.

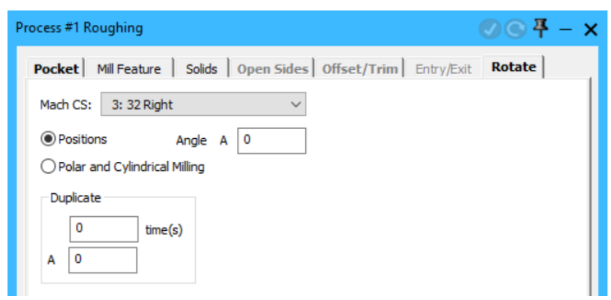
1. Create this Roughing process with the Face Mill.

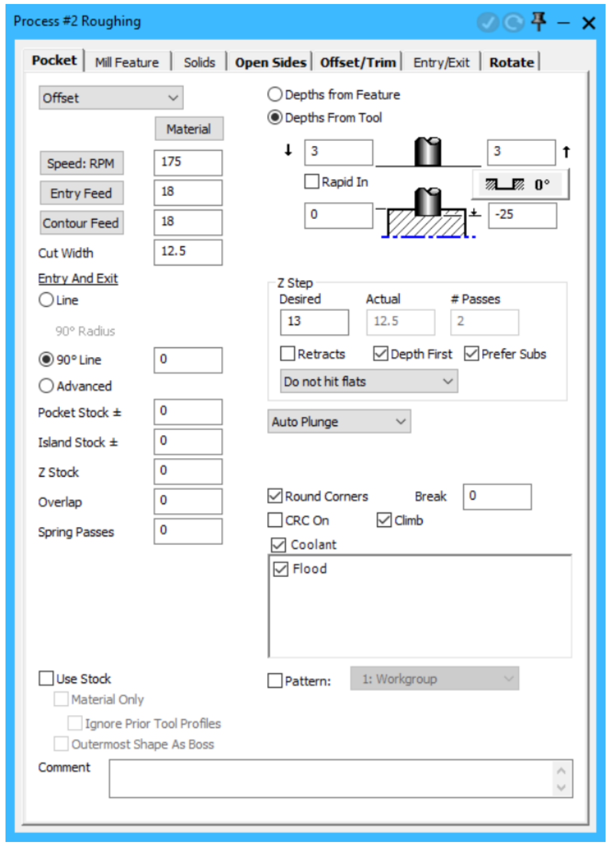


We have designated a material height of 65mm because the stock material is well above Z0. This will ensure we do not violate the part. The actual stock extends to a little more than 63mm above Z0.

2. Set the Mach CS option in the Rotate tab to 32 Right.

   Because the face milling roughing option allows you to face off the entire stock, we can combine the pocketing operation with the roughing operation in the same Process list.

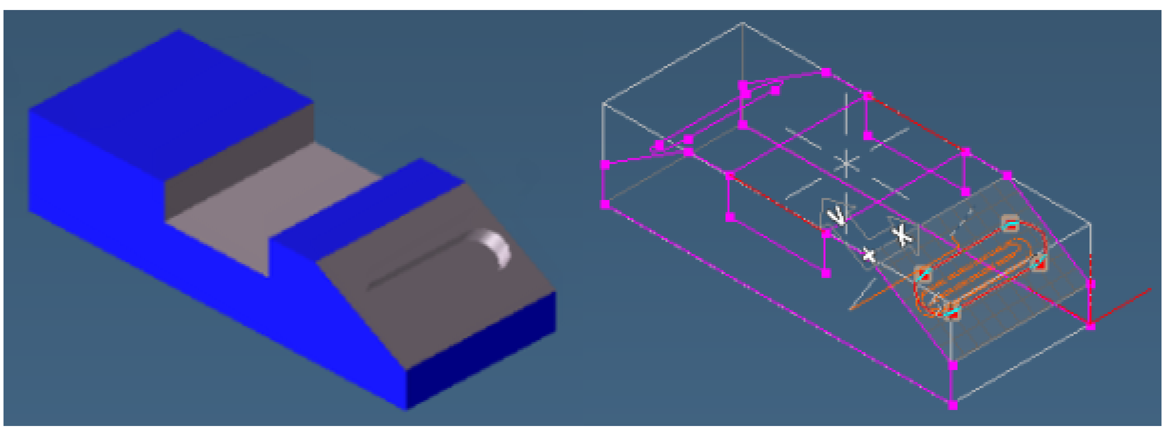




3. Create this Roughing process with the Rough Endmill.

   The settings in the Rotate tab do not need to be checked. Simply setting this option once in a process will affect all other processes in the list.

4. Select the slot geometry and create the operation for the slot.



Don’t forget to deselect the previous operation before creating this toolpath.

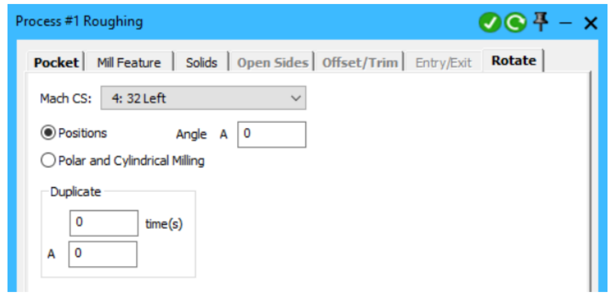
5. Render the part.

Operations 4 and 5, 32° Left Face

Now, we will machine the other 32° plane. We can use the same process group to machine the other side.

1. Deselect the operation tiles.

2. Change the Mach CS in the Rotate tab to 32 Left.

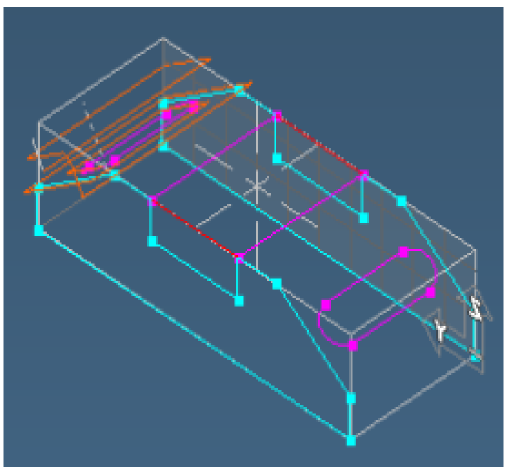


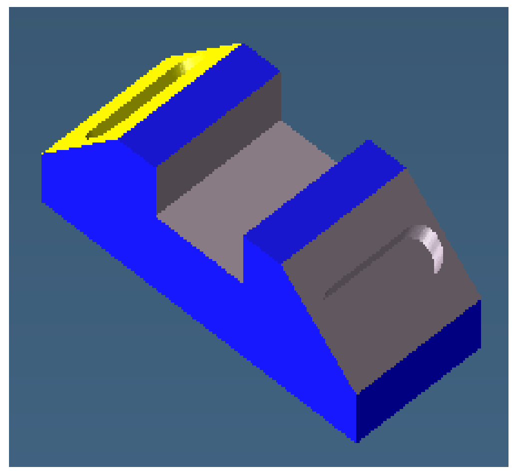
All the other information in process dialogs should remain the same. The other processes in the group will all change their CS's as well.

3. Select the other slot geometry for the left side and create the toolpath.
4. Render the part.

Note that you did not have to switch coordinate systems.

All of the operations to cut the part have been created.





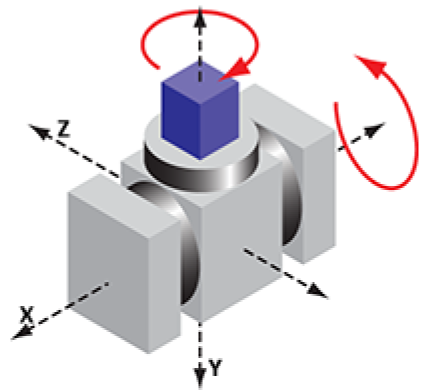
5-Axis Part: Hinge

About the Hinge Part

To see the coordinate system construction and geometry creation for this exercise, see "5-Axis Part: Hinge" on page 30. For the part print for the exercise, see "Hinge" on page 107.

Setting up the Hinge

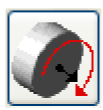
1. Open the Hinge.vnc part file.

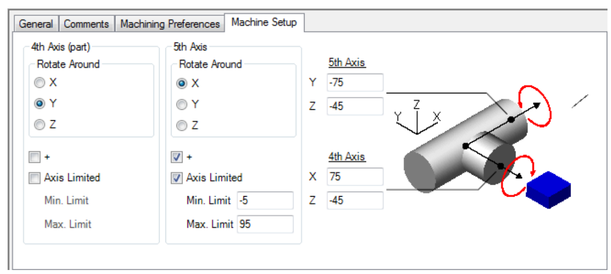


You may recall that the part is 150x75x90mm (X+150 X-0, Y+75, Y-0, Z+0 Z-90. This part will be programmed for a 5 Axis Horizontal Mill. The part is gripped from the front (Y- side). In another situation we would probably leave more material on the Y- side of the part so that the part can be cut off, but for greater visualization of the part we are not including the extra material that would get cut off.

Rotary Setup

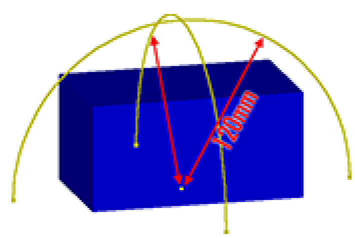
1. Open the Document Control dialog.

2. Click the Rotary Setup button  and enter the information in the 5 Axis Setup dialog as shown.



The part is oriented so that the 4th axis is centered in X and Z while the 5th axis is centered in Z and 75mm below the bottom of part.

3. Set the part's Clearance Plane to 120mm.



It is a little more than 106mm from bottom-center of the part (where the machine will rotate) to a top corner. 120mm will give us adequate clearance when the part moves.

Tool Creation

1. Create the following tool list:

#	Type	Total Length	Diameter	Corner/Tip	# Flutes	Flute Length	Material
1	rEM	75mm	75mm	13mm	2	50mm	HSS TiN Coated
2	rEM	100mm	26mm	0mm	2	80mm	HSS TiN Coated
3	Shell	75mm	50mm	0mm	4	50mm	HSS TiN Coated
4	Drill	75mm	8mm	118°	2	N/A	HSS

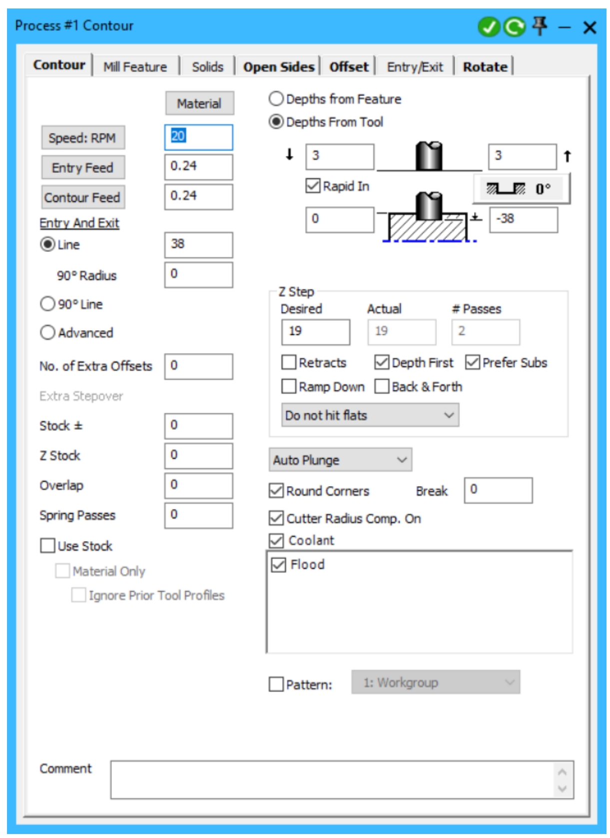
Machining The Hinge

Operation 1

The first operation will contour the front side of the part. We will use the rough endmill with the 13mm bottom corner radius to create the necessary fillet.

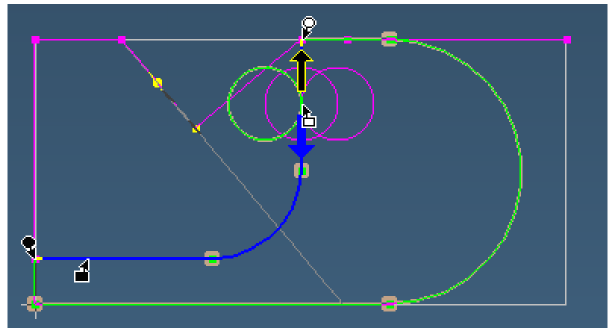
1. Create this Contour process with tool #1.

   The cut depth of -38mm will cut to the center wall of the part. With an Entry Move and Exit Move of 38mm the tool will plunge down, off of the part by just more than its radius, then feed into the part. We do not need to enter any data in the Open Sides or Rotate tabs.

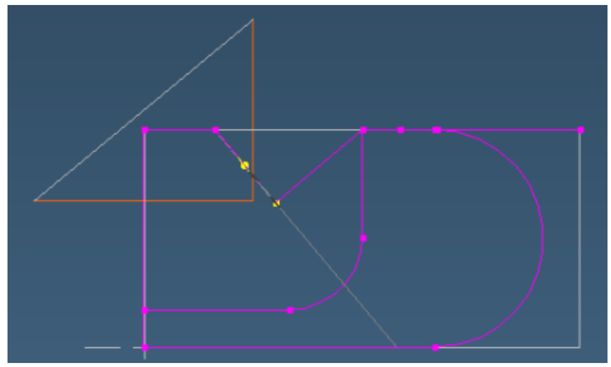


2. Select the cut shape by positioning the machining markers as shown.

   The start and end point markers are placed on the connecting points at the edge of the part.

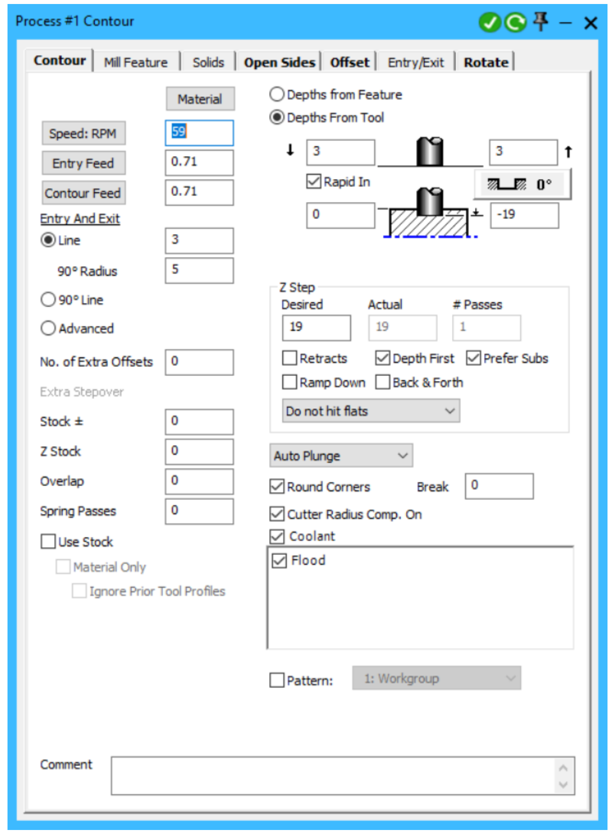


3. Click the Do It button or use (Ctrl+.).



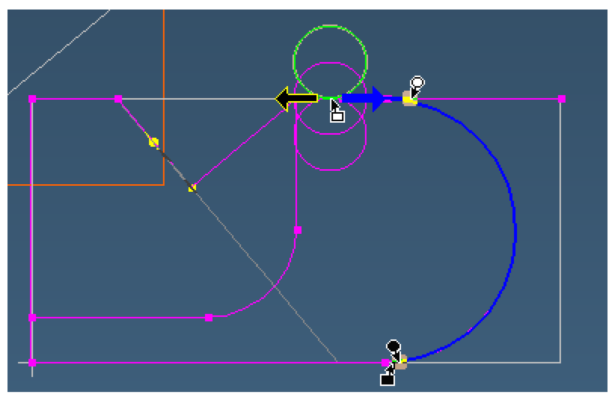
Operation 2

Next, we will contour the other portion of the front side profile.

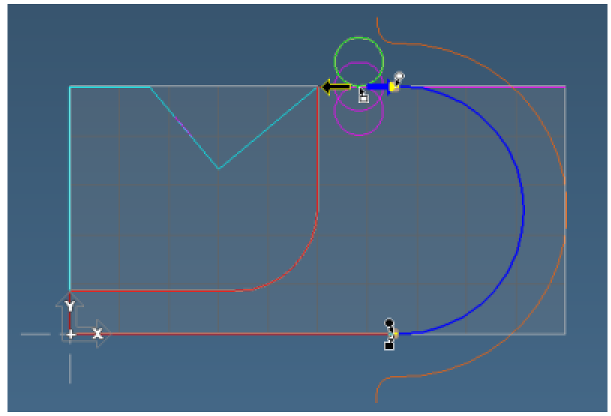


1. Create this Contour process with tool #2.

   Note that we have a line and radius value applied to the entry and exit moves. Again, we do not need to set anything in the Open Sides or Rotate tabs.

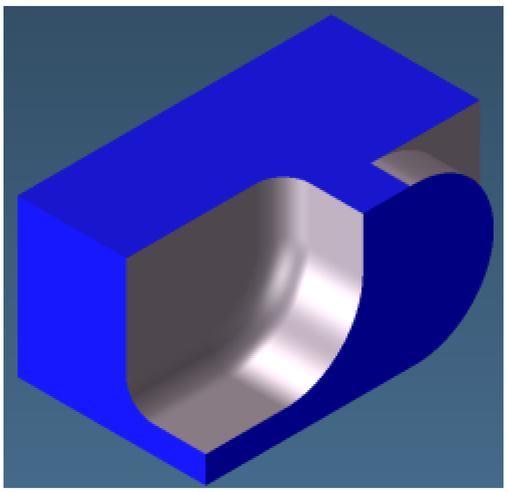


2. Select the cut shape by positioning the machining markers as shown.



3. Click the Do It button or use (Ctrl+.).

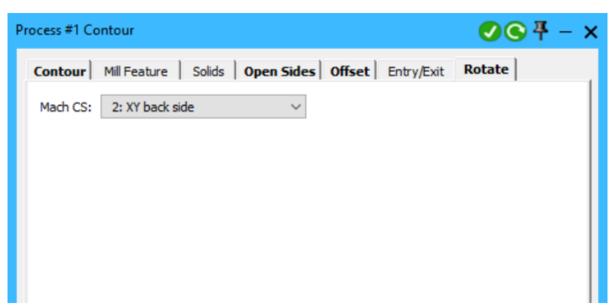
4. Render the operations.



Now, we will contour the back side of the part. The processes will be identical to the first two operations, except for the machining CS and the selected cut shape.

Operation 3

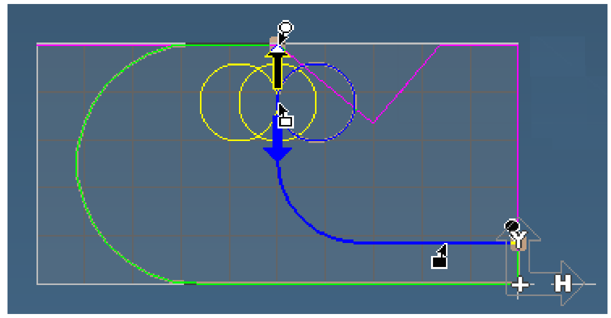
1. Switch to the XY back side CS.
2. Change to the home view (Ctrl+H).
3. Double-click operation #1 to reload it.



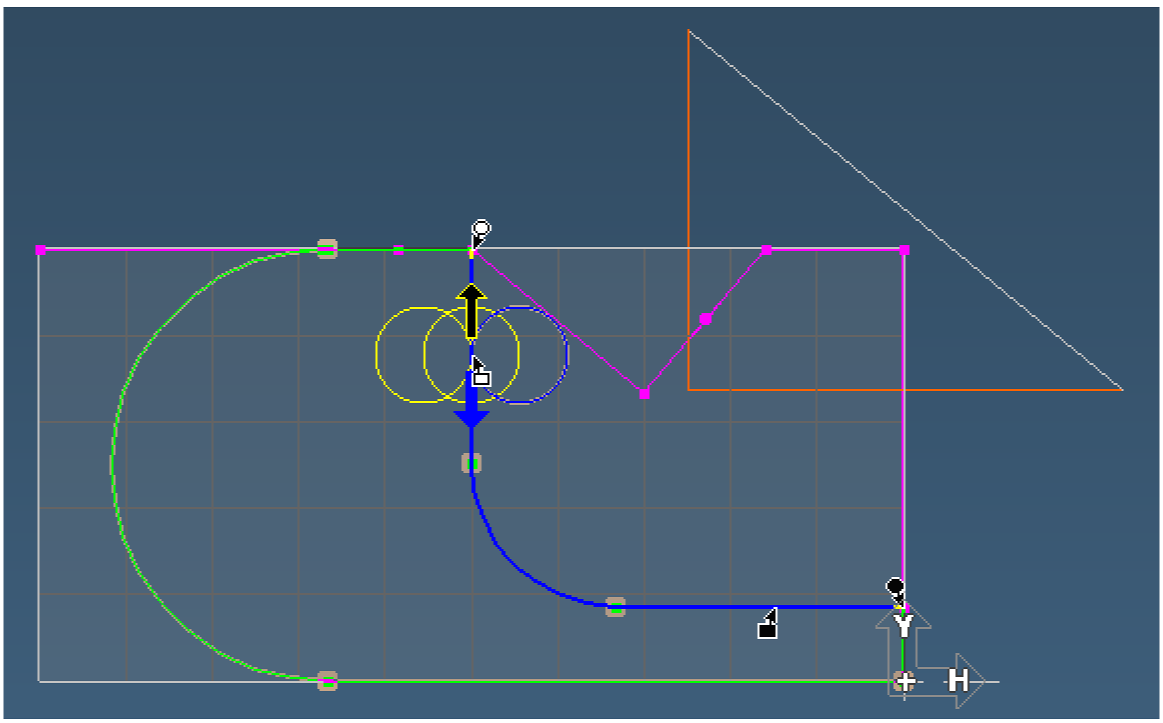
4. Open the process dialog and change the Mach CS in the Rotate tab to XY back side.

You will notice that the only available item on this tab is the Machining CS. This is because the the setup in the DCD for 5-axis makes it so that the rotary axis is shifted off the part origin.

5. Select the cut shape by positioning the machining markers as shown.



6. Click the Do It button.

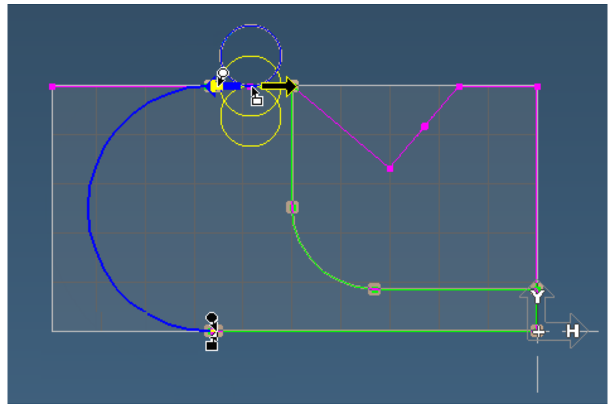


Operation 4

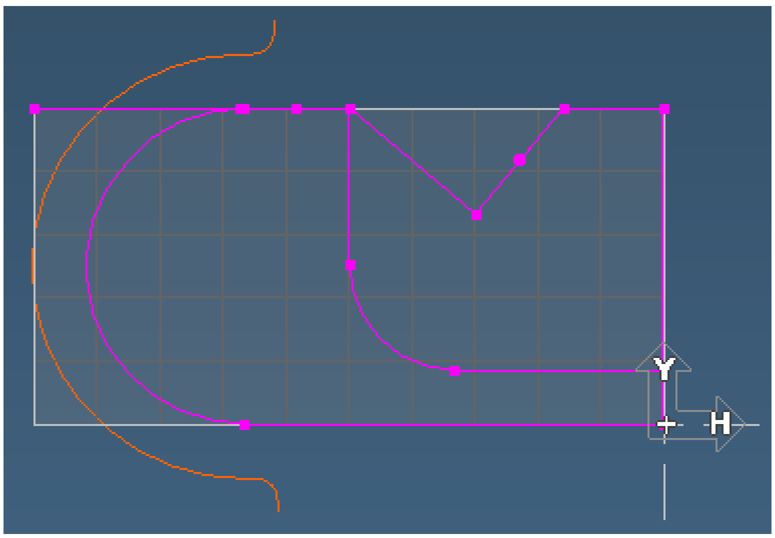
As with Operation 3, the information in the process dialog for this operation is the same as for operation #2, except that the Machining CS must be set to the XY back side CS.

1. Double-click operation #2 to reload it.
2. Open the process dialog and change the Mach CS in the Rotate tab to XY back side.

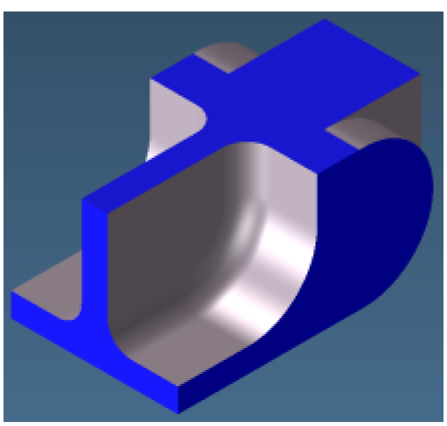
3. Select the cut shape by positioning the machining markers as shown.



4. Click the Do It button.



5. Render the part.

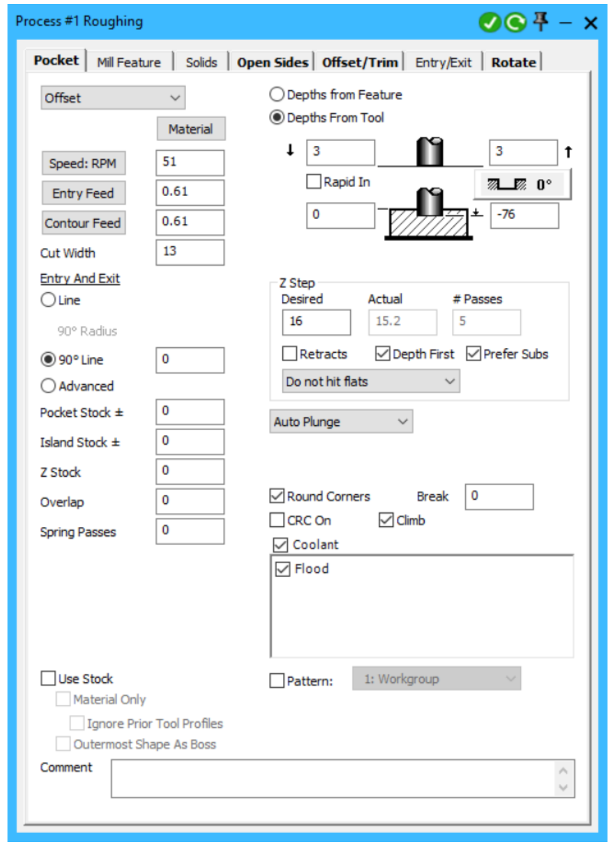


Operation 5

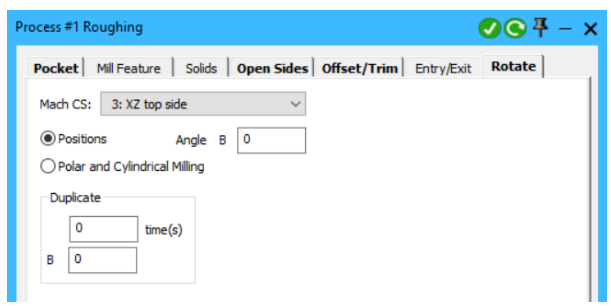
The next operation will pocket out the top side of the part.

1. Switch to the CS, XZ top side.

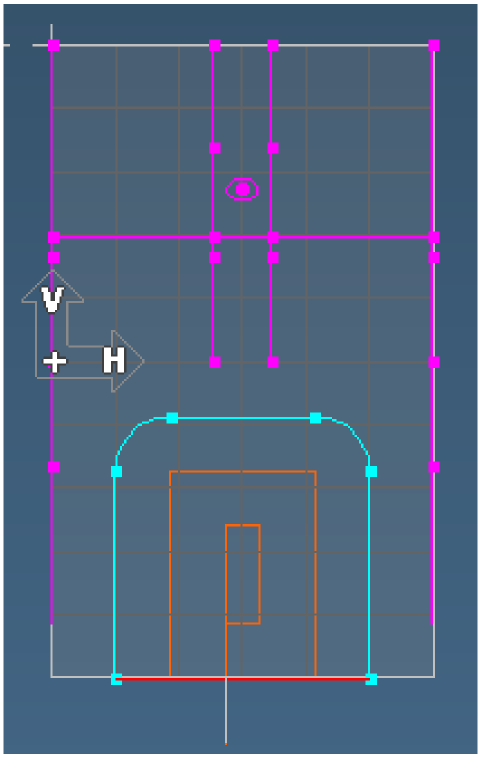
2. Create this Roughing process with tool #2.

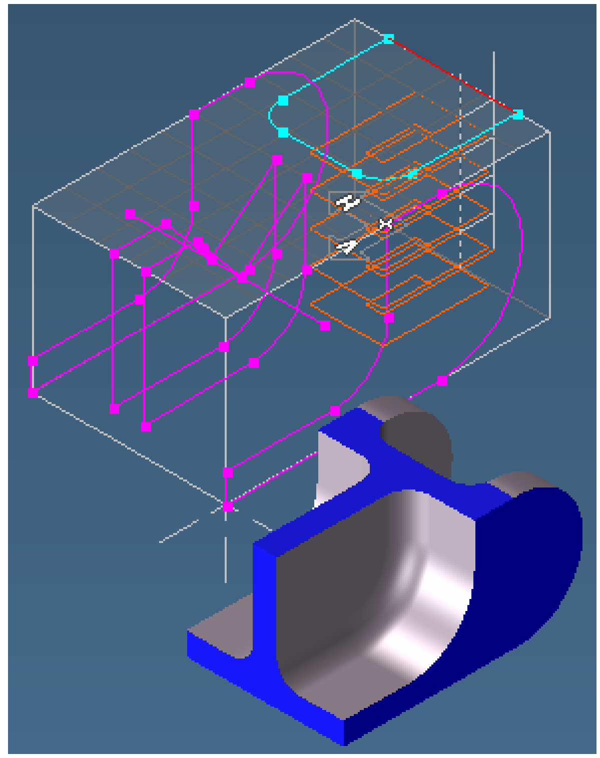


3. Set the Mach CS to XZ top side.



4. Select the pocket geometry, `click` the Do It button or use (`Ctrl+.`) and render the operations.



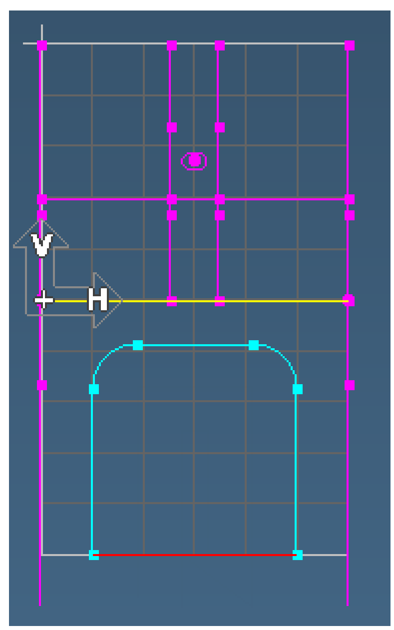


Operation 6

In order to complete the part, we must generate operations to create the angle on the center wall and drill the hole.

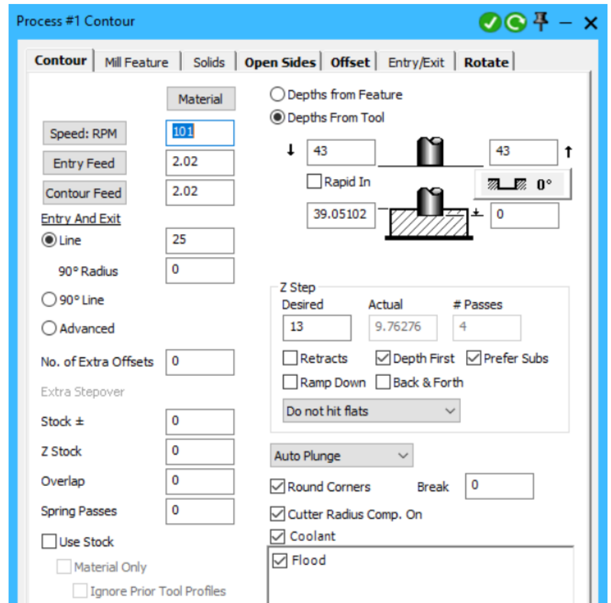
1. Switch to CS4: Hole and change to the home view (`Ctrl+H`).

   First, we need to create a line in this CS so that we can create a contouring operation which will create the angled wall.



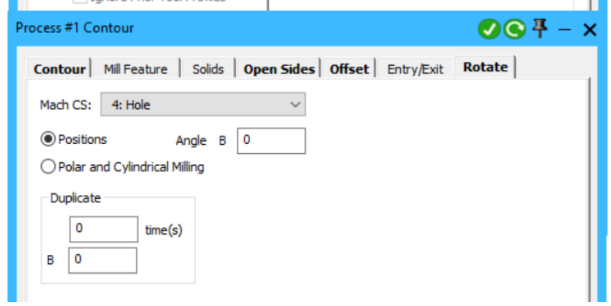
2. Create a horizontal line at V=0.

   The line should be terminated at each end so that we can position the start and end point machining markers.

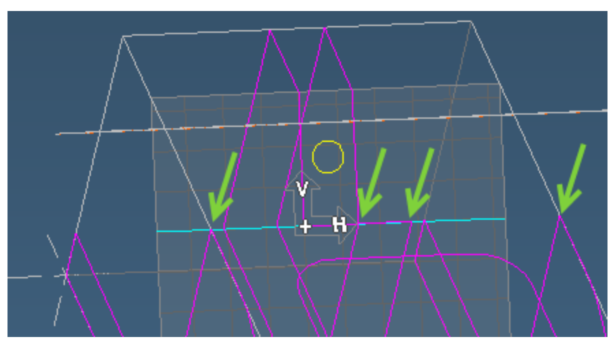


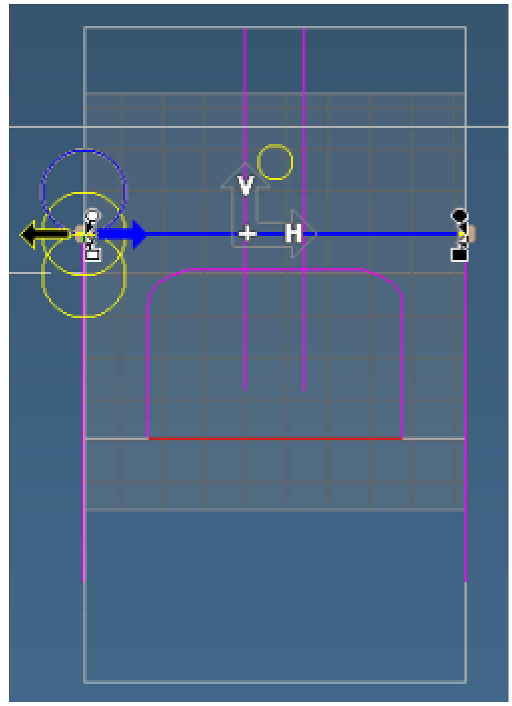
3. Create this Contour process with tool #3.

4. Select Hole as the Mach CS.



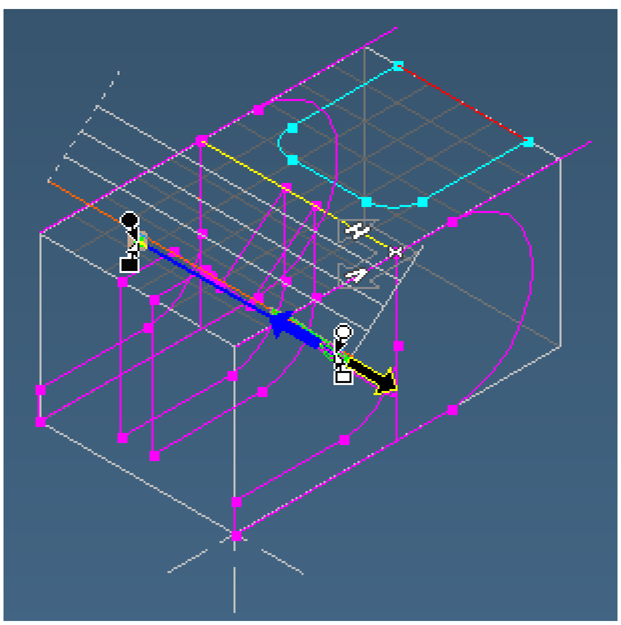
The 25mm line gives us room for the tool to feed onto and off of the part. The Surface Z value of 39.05102mm is acquired by interrogating any of four points.



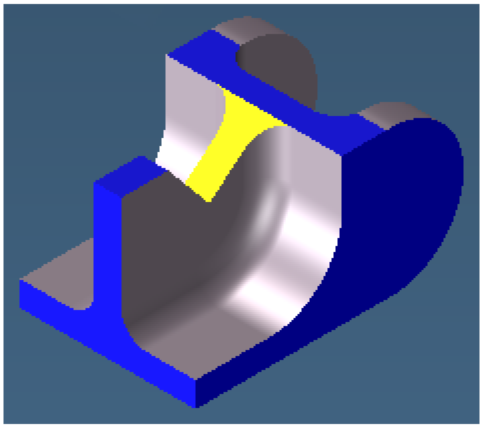


5. Select the new line and position the machining markers as shown.

6. Click the Do It button.



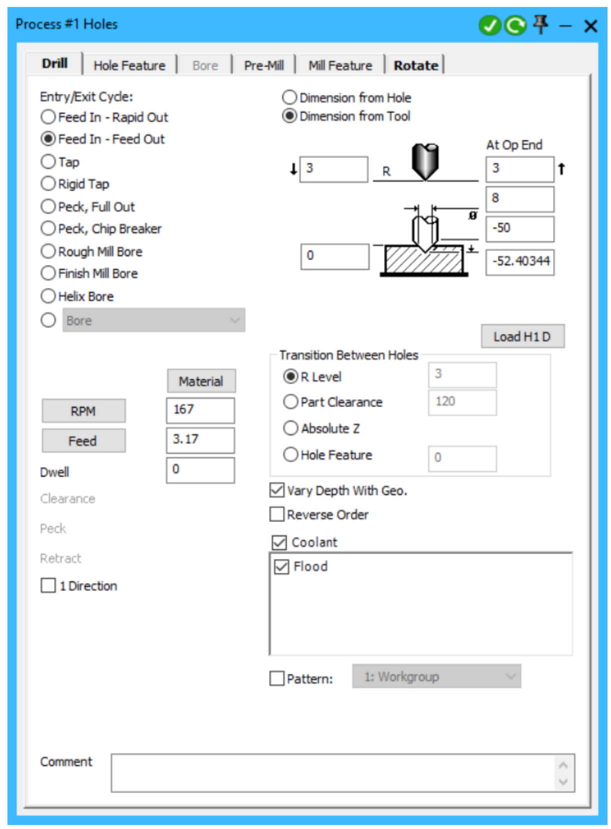
7. Render the operation.



Operation 7

The final operation drills the hole in the angled wall.

1. Create this Holes process with tool #4.



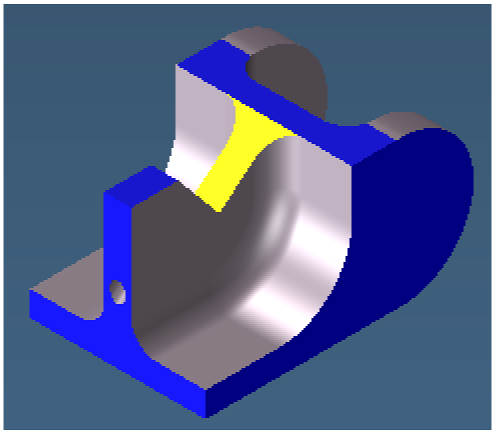
2. Select Hole as the Mach CS as the machining CS.

   A full diameter depth of -50mm will drive the tool through the left side of the part.

3. Select the geometry for the hole and `click` the Do It button or use (`Ctrl+.`).

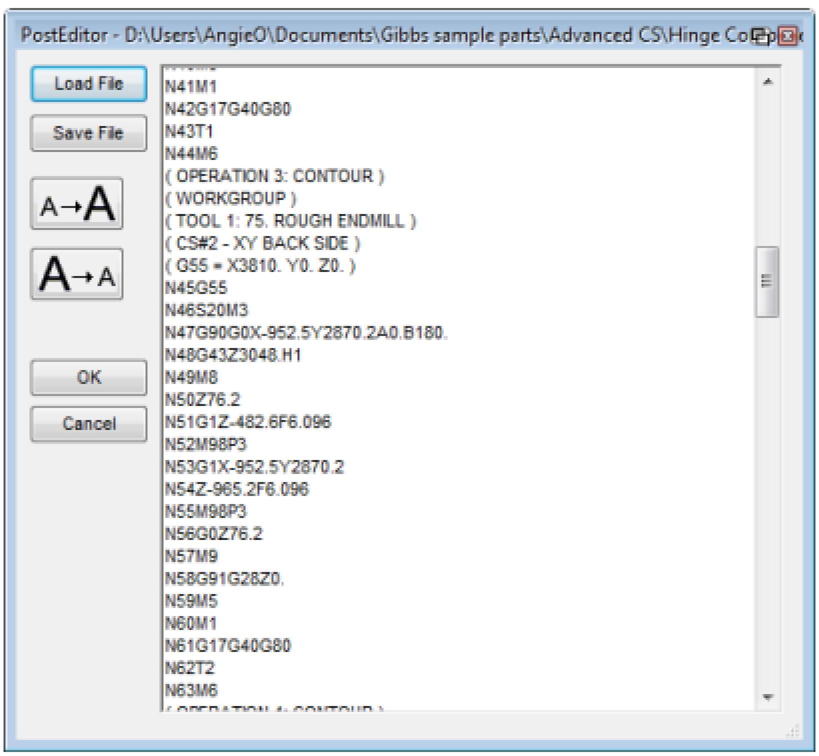
   All the operations to program the part have now been completed.

4. Render the operations.



Note that the hole goes through the left side of the part.

Normally, fourth and fifth axis rotary parts are processed using a C-style post processor so that the rotations are output in the code, rather than as a WFO. An example of the output is shown here.



5. Save this part.

WORK FIXTURE OFFSETS

We have seen how multiple coordinate systems allow us to manipulate the part and machine it. Another use for multiple coordinate systems is to utilize work fixture offsets (WFO) to machine multiple coordinate systems and/or multiple sides of parts in a single program. In order to use work fixture offsets, parts must be post processed with an Advanced CS Post Processor, typically a B-Style post.

This chapter contains two exercises which use WFOs. The first creates two parts, a left and right, and machines them using WFOs. The second exercise machines the top and bottom sides of a part in a single program.

When using work fixture offsets, the system matches each coordinate system with a WFO. This is done in the order of the coordinate systems in the CS list dialog. CS1 is always WFO 1 (G54); CS2 is always WFO 2 (G55); CS3 is WFO 3 (G56); etc. This can be overridden by specifying a WFO number in the CS list. For more information on WFOs, refer to the Advanced CS User guide, "Rotary Tables/ Work Fixture Offsets" chapter. By assigning a specific WFO number to a CS you can more easily create and modify your part.

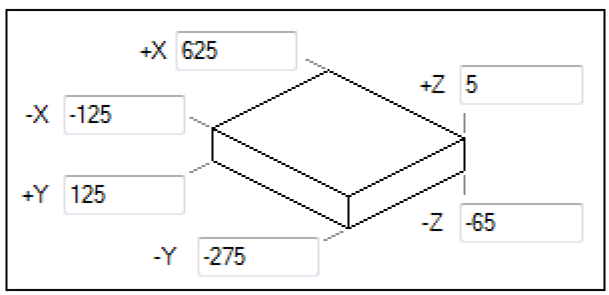
Family of Parts

In this exercise, we will create geometry for the left side of a part, mirror the geometry to create the right side of the part, and then machine both parts in a single program using different machining coordinate systems for work fixture offsets. The left side will be done in G54 and the right side in G55. For the part print for this exercise, see “Family of Parts” on page 108.

Making the Model

Part Setup

1. Create a new metric part with a 3 Axis Vertical Mill MDD with the dimensions as shown and a Clearance Plane value of 15mm.



Tool Definition

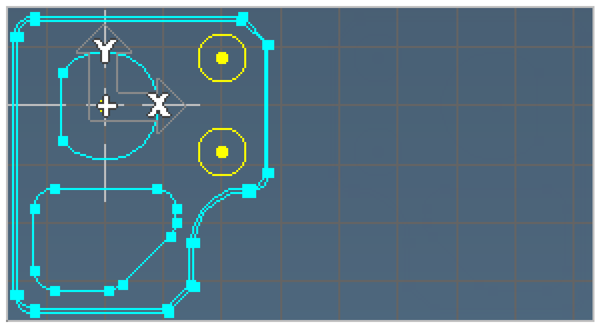
1. Create the following tool list:

#	Type	Total Length	Diameter	Corner/Tip	# Flutes	Flute Length	Material
1	Face Mill	63mm	160mm	1mm	5	15.5mm	HSS TiN Coated
2	fEM	166mm	25mm	0mm	3	90mm	HSS TiN Coated
3	Rough EM	123mm	16mm	0mm	3	63mm	HSS TiN Coated
4	Drill	100mm	40mm	118°	2	N/A	HSS TiN Coated
5	fEM	123mm	16mm	0mm	3	63mm	HSS TiN Coated

Geometry Creation

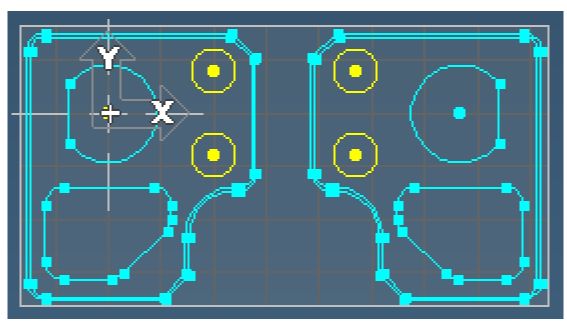
1. Create the part geometry with the center of the D-hole at the origin.

   You only need to create the outer profile of the walls for the part, but you will also need to create the inner profile of the exterior wall for machining. The Plug-ins > Create DHole will create the D-hole but remember to add the radius to the flat diameter (125) to get the proper flat distance.



To create the geometry for the right side of the part, we will duplicate and mirror the left side geometry.

2. Select all (`Ctrl+A`) of the geometry.

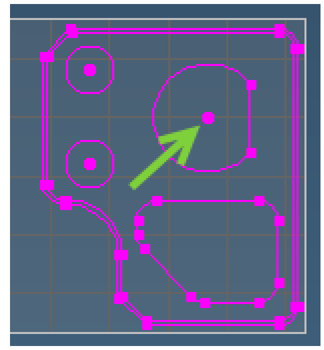


3. Choose Modify > Duplicate And... Mirror the geometry about the X axis at `250`mm.

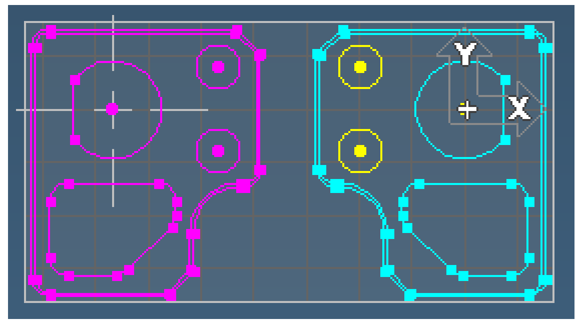
G55 Coordinate System

Now, we will create a new coordinate system for the right side of the part.

1. Create a new CS and name it `G55`.



2. Select the point shown and `click` the Change CS Origin  button.



3. (`Shift+drag`) to select the mirrored geometry and choose Modify > Change CS (XYZ).

   The geometry will remain in the same location, however it is now defined in CS2 and is drawn in blue and yellow rather than magenta, indicating that it is based on a different coordinate system.

Machining the Part

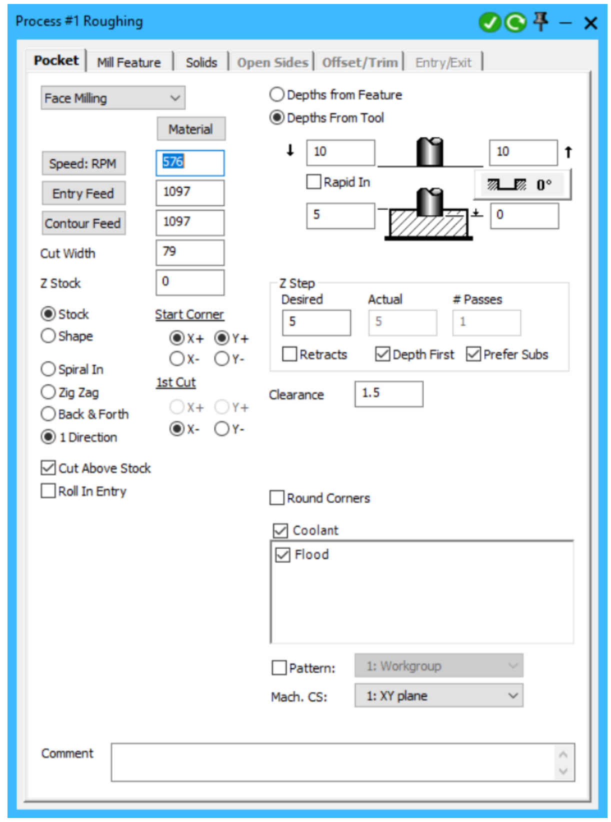
The geometry and coordinate system creation is complete. Now, we will create machining operations to cut both sides of the part.

Face Mill

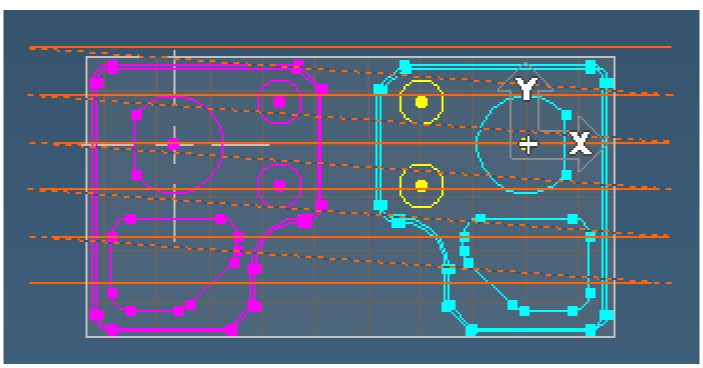
First, we will create an operation to face mill the top of the part. We will face off the entire stock in one operations which will encompass both sides of the part.

1. Switch to CS1.

2. Create this Roughing process with tool #1.

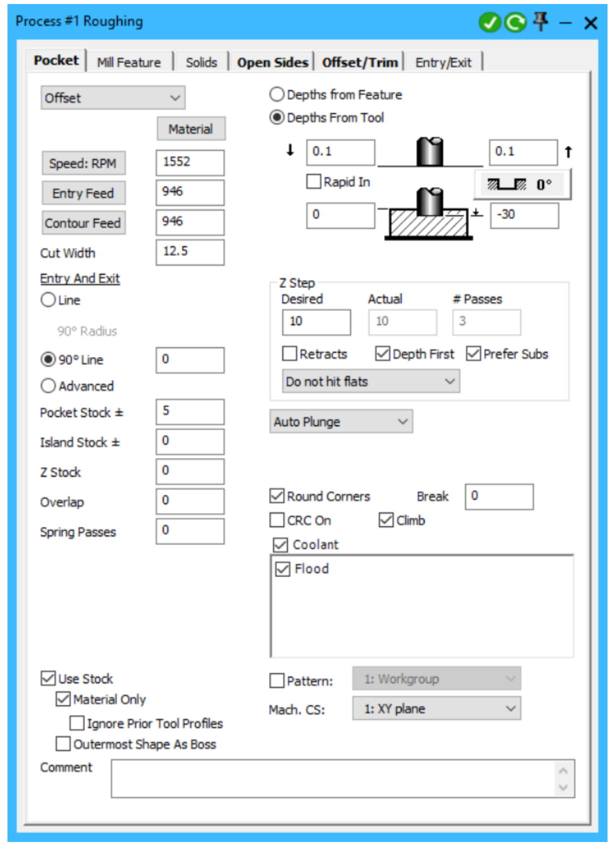


3. Click the Do it button.



Deep Pockets

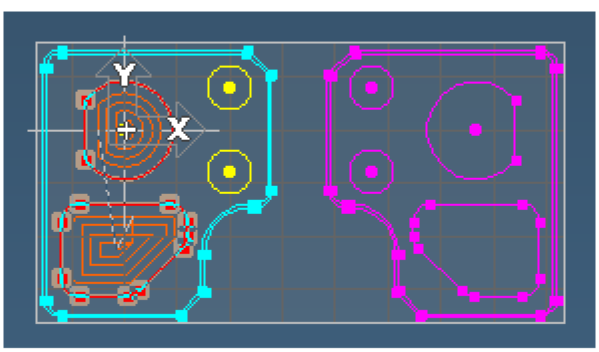
Next, we will create operations to pocket out the D-hole and slot. We will use separate process groups for each side of the part. This allows us to set a different machining CS (G55) for the right side operations. In the posted output, the left side operations will use G54 and the right side operations will use G55. The work fixture offsets will contain the origin shifts for each coordinate system.



1. Create this Roughing process with tool #2.

   Make sure that the Mach CS is set to the XY plane.

2. Select the D-hole and slot geometry on the left side part and `click` the Do it button.

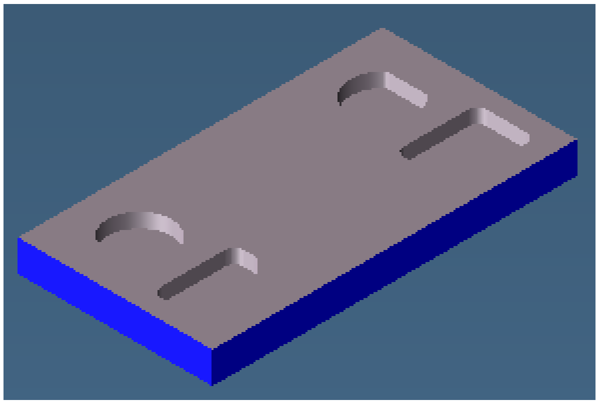


Deep Pockets - G55

The same process can be used for the right side pockets.

Make sure the operation tiles are deselected in the Operations list.

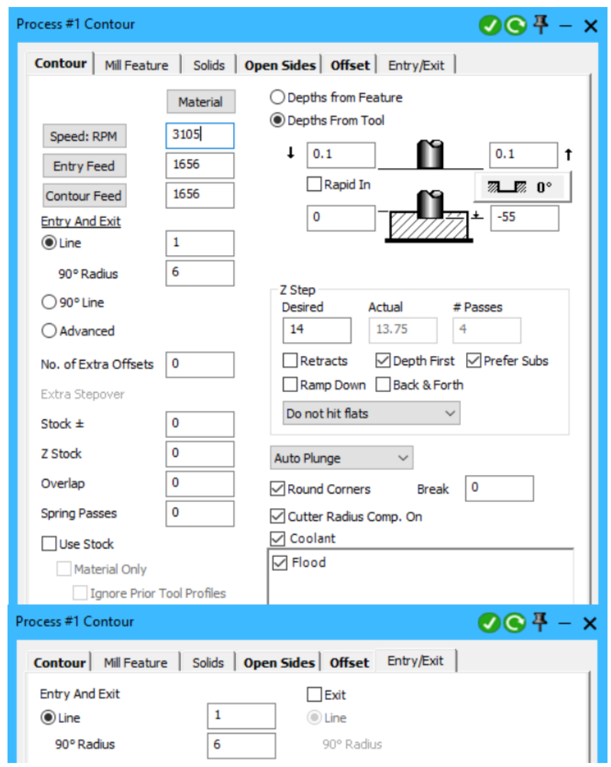
1. Change the Mach CS to G55.
2. Select the D-hole and slot geometry on the right side part and `click` the Do it button.



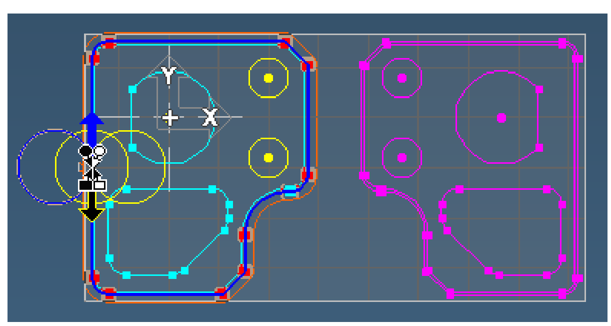
Contour

Next, we will create operations to contour the outside profile of each part.

1. Create this Contour process with tool #2.



2. Position the machining markers as shown.

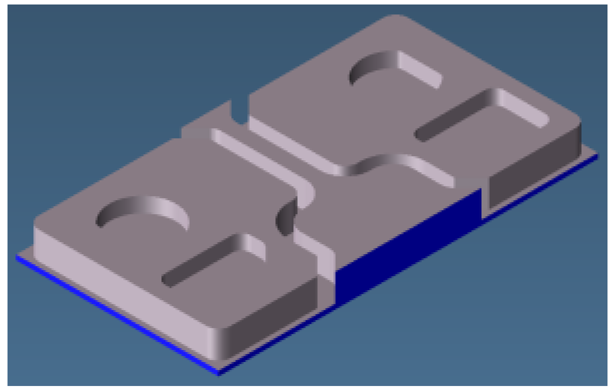


3. Click the Do It button.

Contour G55

1. Use the same Contour process with G55 selected for the Mach CS. Select the outside contour of the right side part for the cut shape. Click the Do it button.

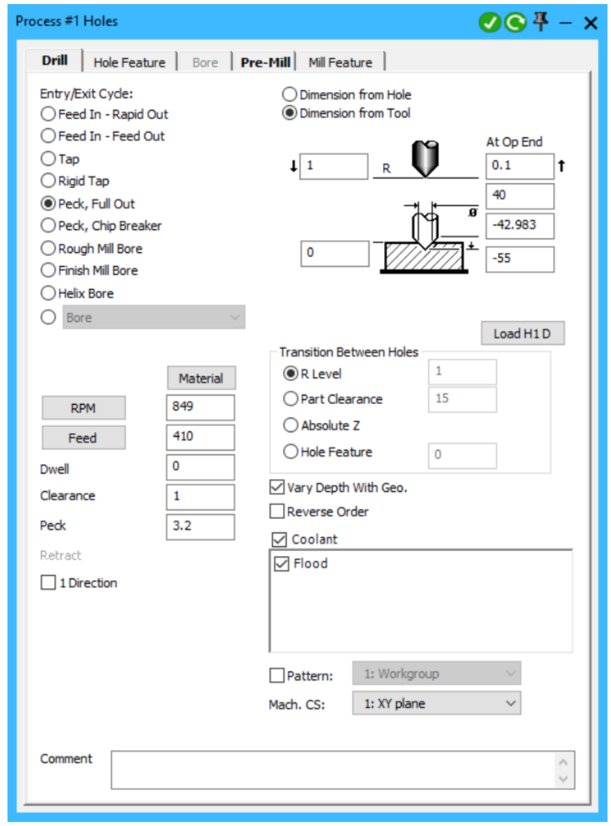
The cut part rendered image should look like the image to the right.



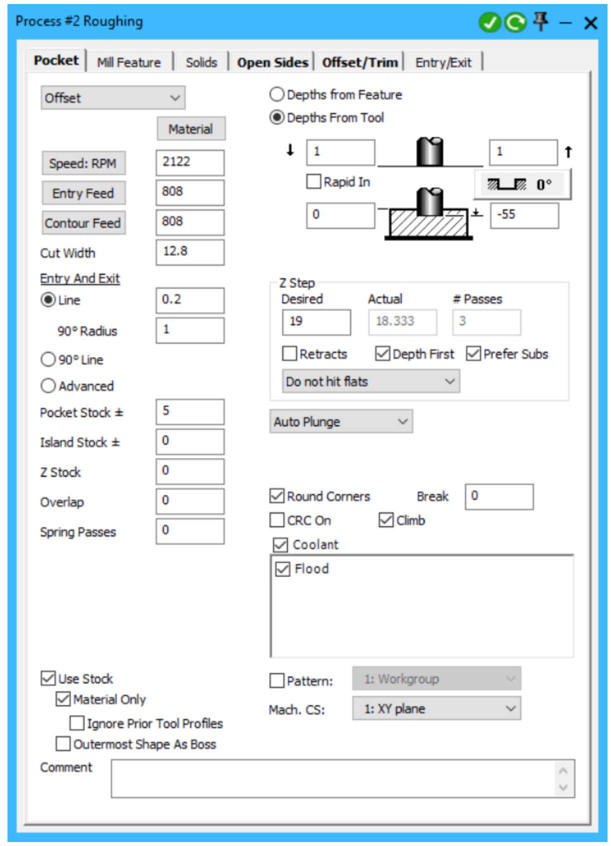
Thus far we have created the operations in the relative order that we want them to be machined. We have alternated between machining the left side and right side, using G54 and G55, to minimize the tool changes. Another method would be to create all the operations to machine the left side part and then create the operations to machine the right side. Once all of the Operation tiles have been created in the Operations list, they can easily be arranged into optimal order for the most efficient machining.

For the remainder of the exercise, instruction will only be given on creating the operations to machine the left side of the part. When you've created the left side operations, use the same process information to create the operations for the right side of the part. Just remember that you need to change the Mach CS to the G55 coordinate system in the process dialog for the right side operations. The next set of operations will drill and pocket the two holes.

Holes

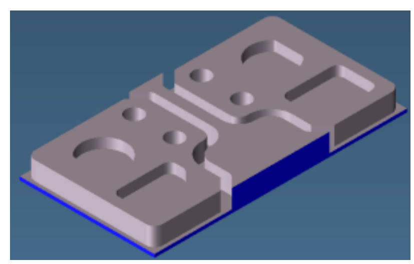


1. Create this Holes process with tool #4.



2. Create this Roughing process with tool #3.

3. Select the two circles and click the Do it button.

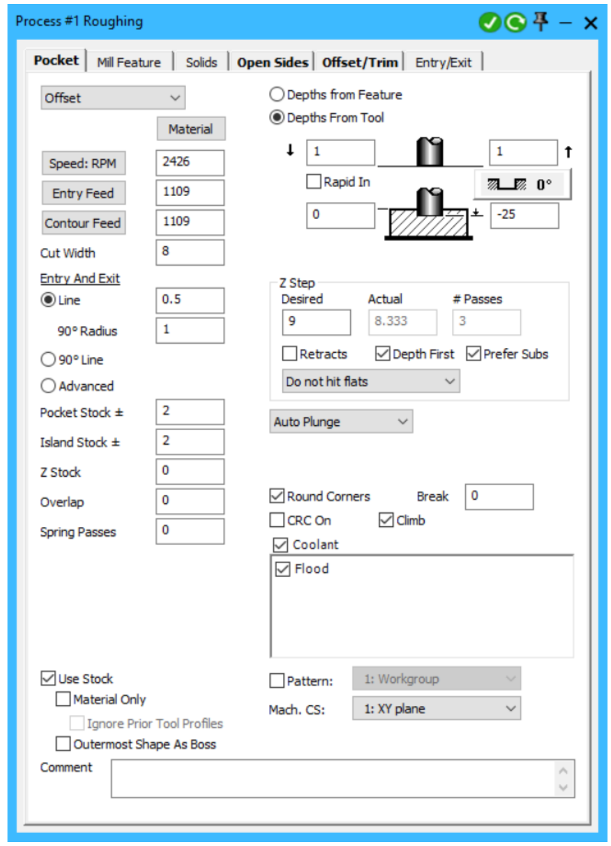


4. Create the drilling and pocketing operations for the right side part. Make sure that you change the Mach CS to G55.

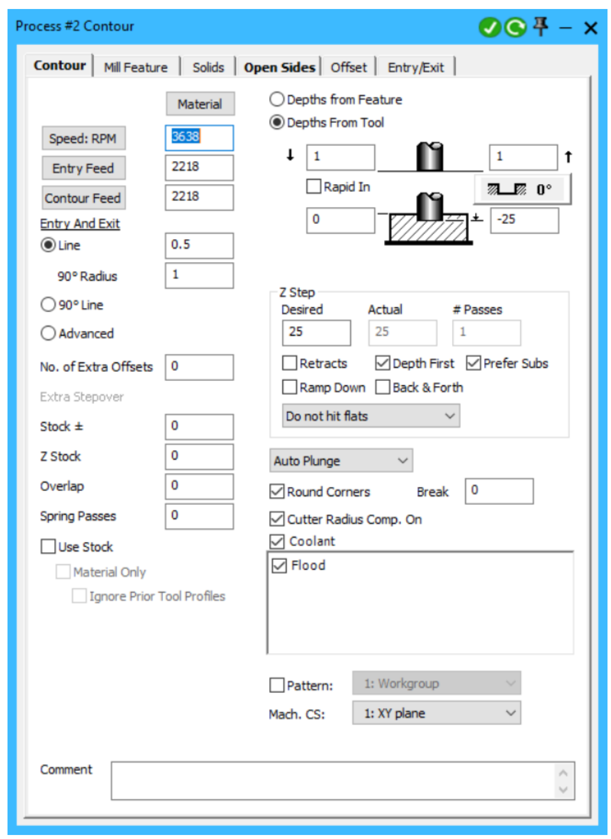
Shallow Pockets

The final set of operations will pocket out the inside of the part, creating the 0.1" wall around each of our pockets.

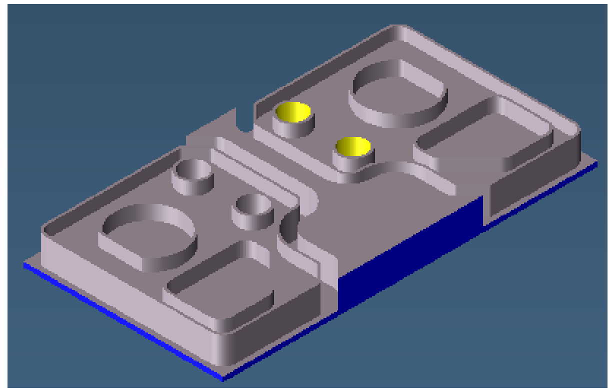
1. Create this Roughing process with tool #3.



2. Create this Contour process with tool #5.



3. Select the outside contour of the part and each of the four pockets inside. Click the Do it button.



4. Create the set of operations for machining the right side part using the same parameters in the process dialogs using the Mach CS: G55.

   All of the operations to machine both the left and right side parts have been completed.

   When using work fixture offsets to machine multiple parts, a B Style Advanced CS Processor should be used so that the code calls G54 or G55 for the offset shifts rather than specifying them in the code itself. This way, the proper origin shifts can be entered at the machine in the work fixture offsets to reflect the actual locations of the parts on the machine.

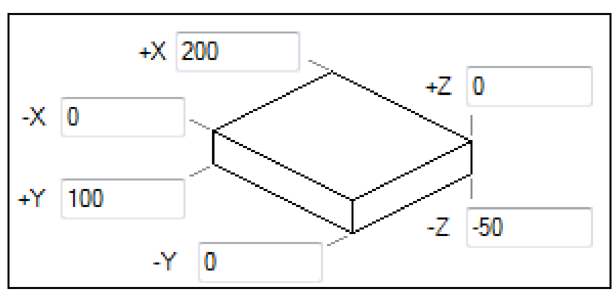
Top and Bottom Machining

In this exercise, we will create a program that will machine the top and bottom of a single part. This is often accomplished using pallets and flipping the pallet for the back side of the part; a manual fourth axis rotation. Because of this, we will program this part for a 4-axis vertical machine, in order to generate accurate code. However, this part is designed to be machined on a 3-axis machine with a manual flip for the back side operations. We will insert a program stop to facilitate the flip. For the part print for this exercise, see “Top and Bottom” on page 110.

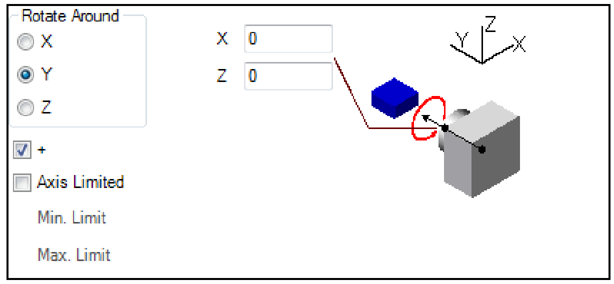
Making the Model

Part Setup

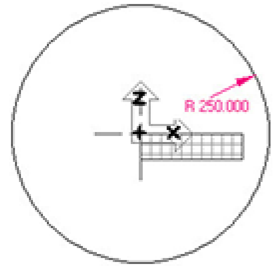
1. Create a new metric part with a 4 Axis Vertical Mill MDD. The part should have stock dimensions as shown with a Clearance Plane value of 250mm. Select a die cast aluminum alloy for the part material if you have the CutDATA material database.



2. Click the Machine Setup tab and enter information into the dialog as shown.

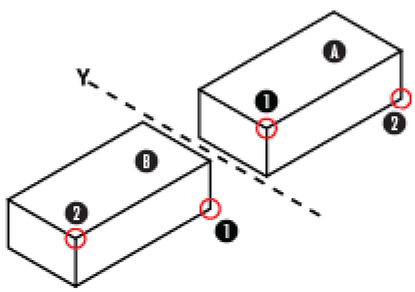


A Master Clearance Plane value of 250mm will give us plenty of room to rotate the part about the Y axis.

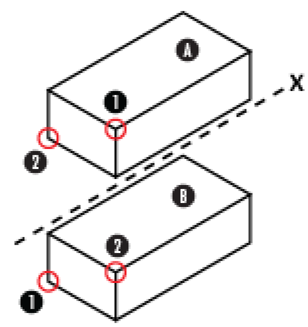


CS Creation

First, we will create a new coordinate system for the back side of the part. This CS will be the XY plane with a different origin. The way we have set up the part, we will be using the Y axis as the rotation axis around which the part will be flipped. The following diagram is intended to illustrate how to visualize where the origin should be for the back side coordinate system.



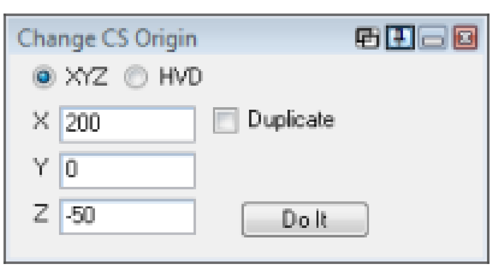
1. Origin 1
2. Origin 2

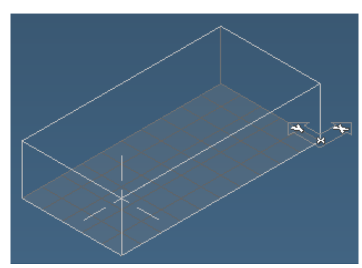


4th Axis Y Flip

4th Axis X Flip

1. Create a new CS and label this CS Back Side.
2. Click the Change CS Origin button in the CS palette and enter the information shown below.





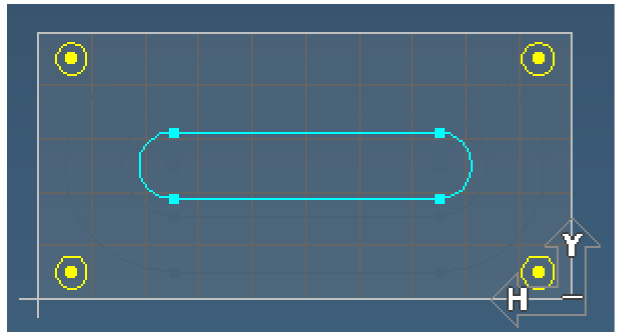
3. Toggle the depth of the CS.

The part's coordinate systems are set. We will now create the part geometry.

Geometry Creation

1. Switch to the Home view for CS2, Back Side.

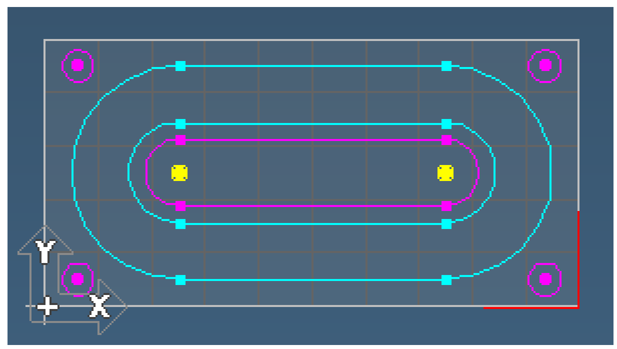
   You can use the part model on the track ball to get an idea of the orientation of this CS in relation to the top of the part indicated by the "T" on the model.



2. Create the geometry for the back side of the part which is drawn in view B-B on the part print.

3. Switch to CS1: XY Plane. Create the top side geometry shown in view A-A on the part print.

   All of the geometry and coordinate systems are now correctly defined. We will now move on to setting up the machining of this part.



Machining the Part

Tool Definition

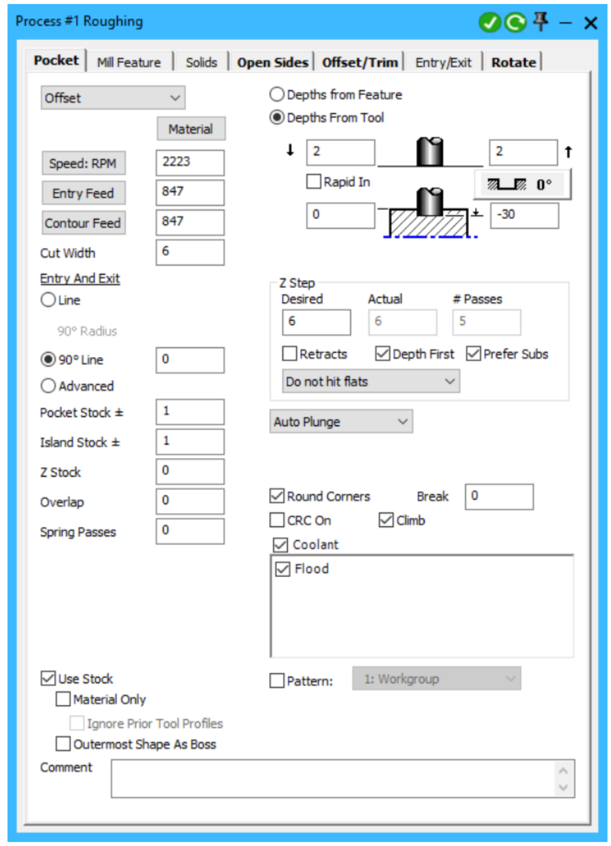
1. Create the following tool list:

#	Type	Total Length	Diameter	Corner/Tip	# Flutes	Flute Length	Material
1	rEM	110mm	12mm	0mm	3	53mm	HSS TiN Coated
2	fEM	110mm	12mm	0mm	3	53mm	HSS TiN Coated
3	Drill	82mm	6mm	118°	2	N/A	HSS TiN Coated
4	Drill	120mm	12mm	118°	2	N/A	HSS TiN Coated

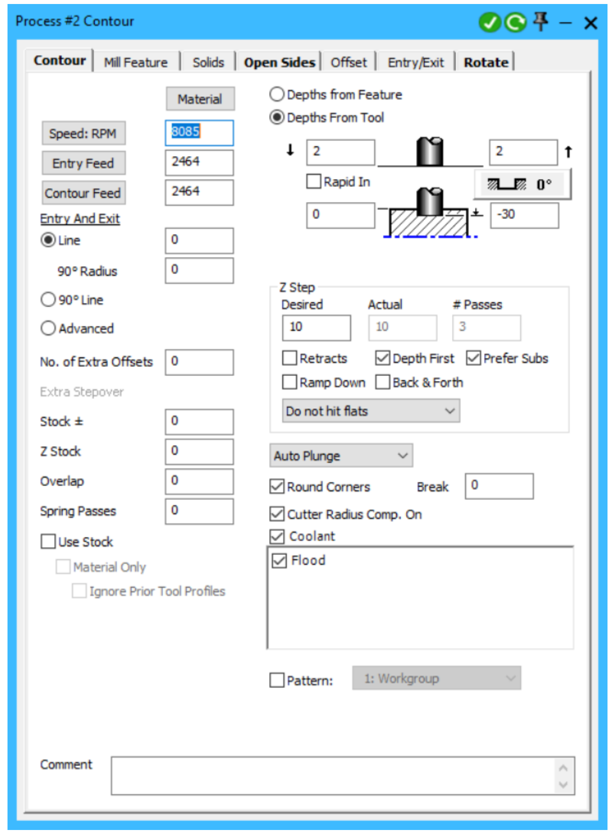
Top Side Pocket

First, we will create the operations to machine the top side pocket.

1. Create this Roughing process with tool #1.

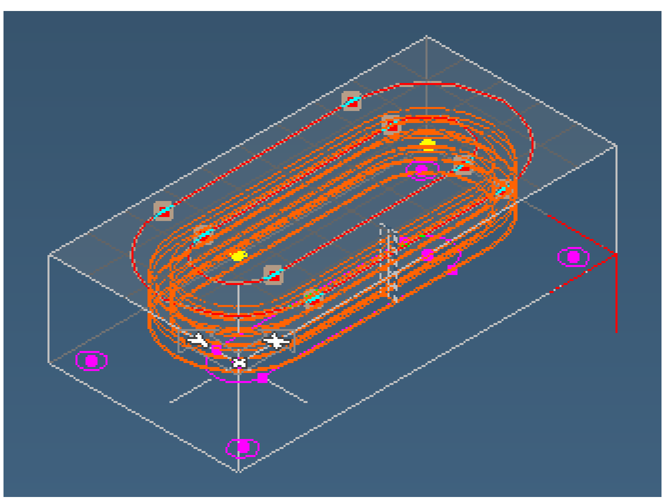


Be sure to select CS 1 as the Mach CS in the Rotate tab.

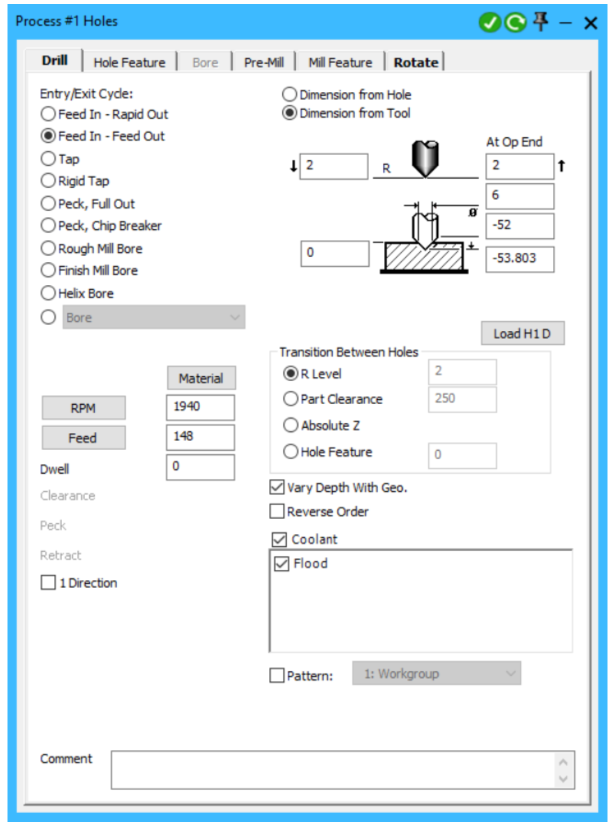


2. Create this Contour process with tool #2.

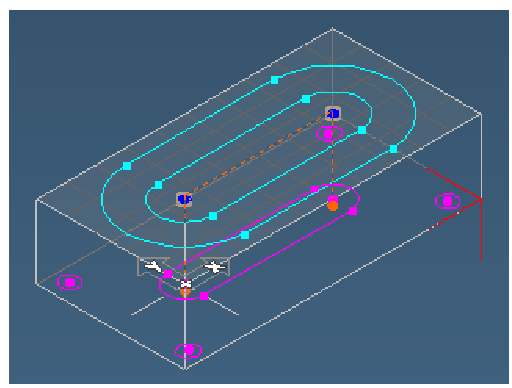
3. Select the pocket and island geometry and `click` the Do It button.



Top Side Holes



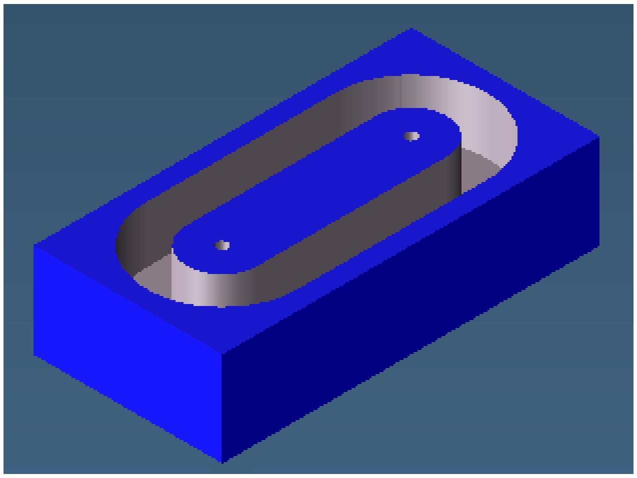
1. Create this Holes process with tool #3.



2. Select the points or circles for the top side drill holes and click the Do it button.

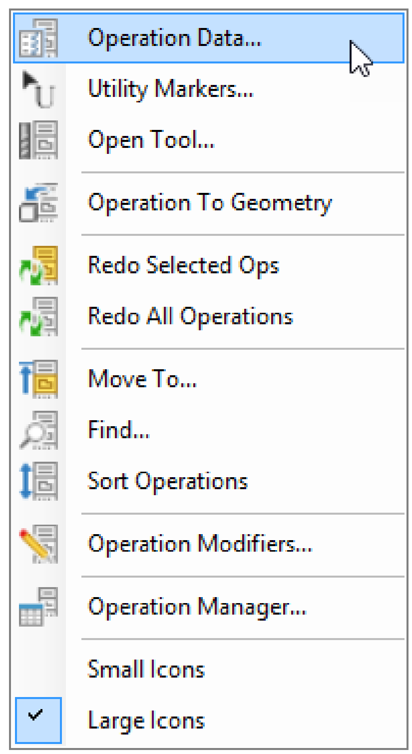
   Holes operations can be applied to either points or circles.

3. Render the operations.



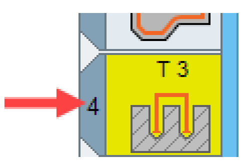
Inserting A Program Stop

The operations to machine the top side of the part are complete. Now, we need to insert a program stop into the posted code before the back side operations. The program stop will allow for the flipping of the part. A program stop can be specified in the Operation Data dialog of any operation.

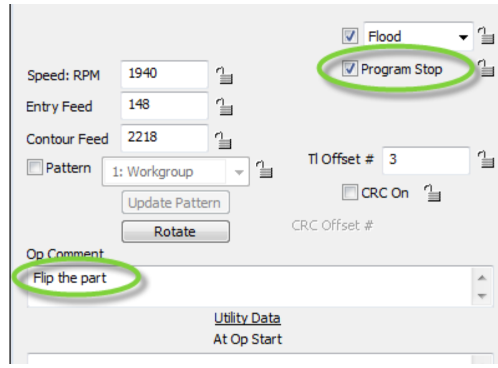


1. `Right-click` the operation tile and select Operation Data.

   Another way to access the Operation Data dialog is by `Double-clicking` on the tab for the operation tile. The operation tab is the item that has the number of the operation on it.



2. Select the Program Stop option in the Operation Data dialog.

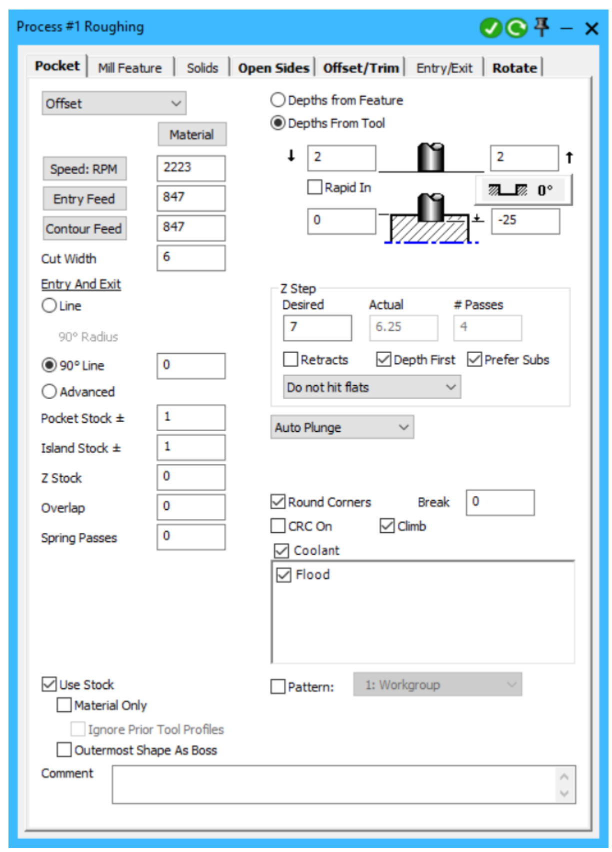


This will create a program stop, usually M0, in the posted code after operation #4 has been completed. Additionally, we inserted an operation comment, `Flip the part`, that will show up in the posted output.

Back Side Pocket

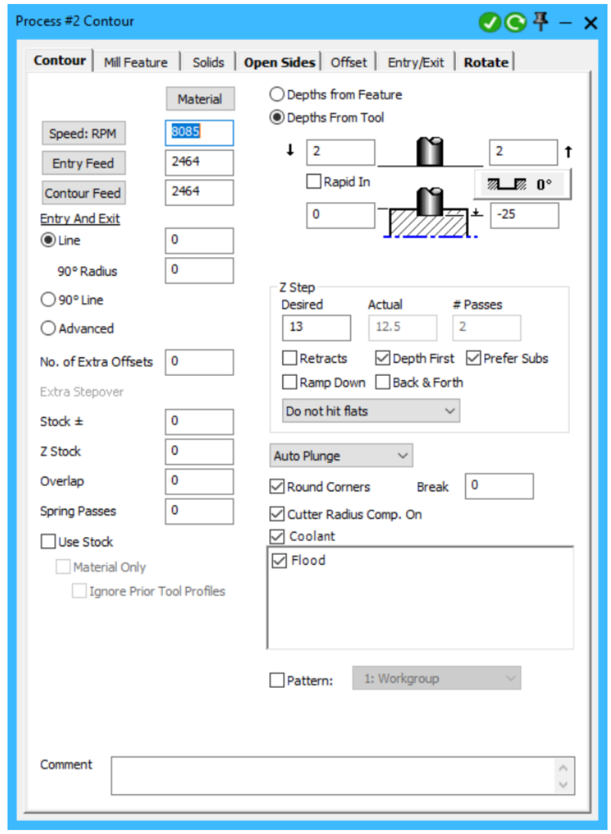
Now, we need to program the operations to cut the back side of the part. We need to make sure the Mach CS is set to the Back Side coordinate system.

1. Create this Roughing process with tool #1.

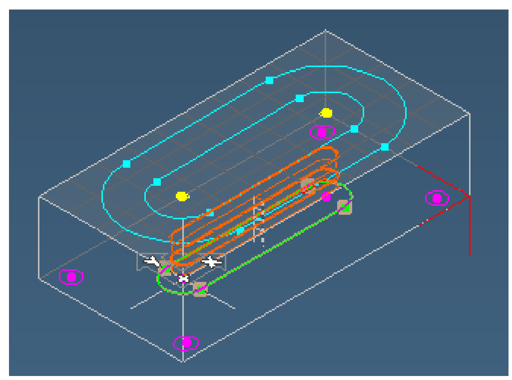


Be sure to select CS 2 as the Mach CS in the Rotate tab.

2. Create this Contour process with tool #2.

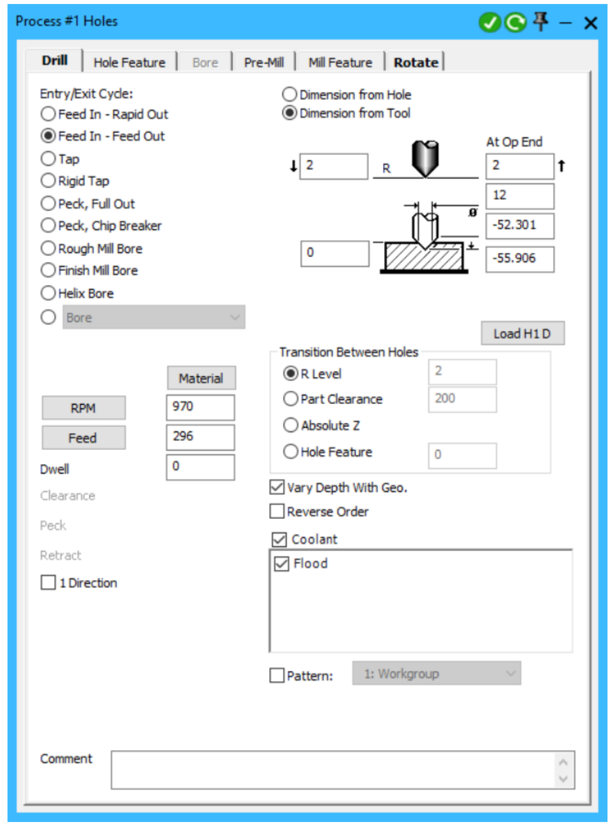


3. Select the slot on the bottom side of the part and click the Do it button.

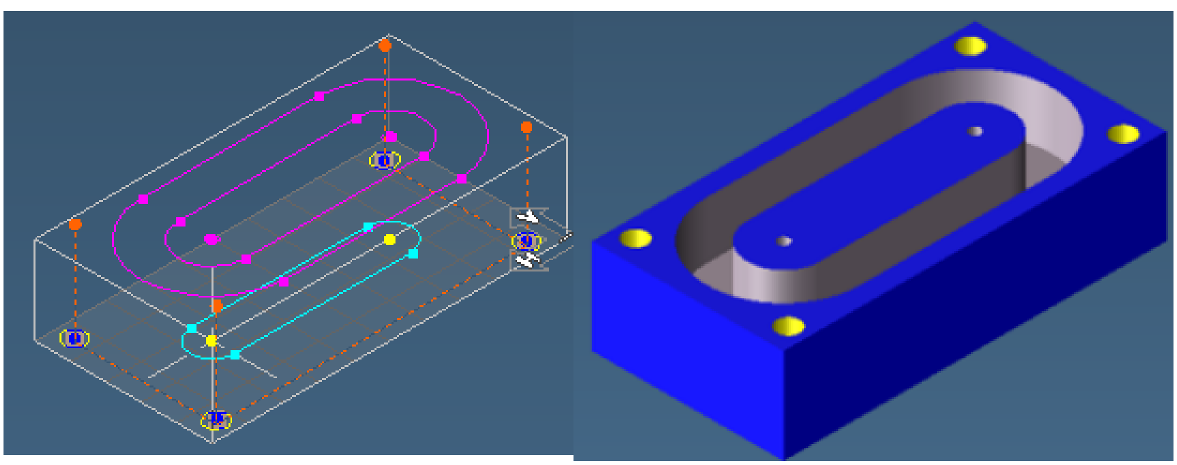


Back Side Holes

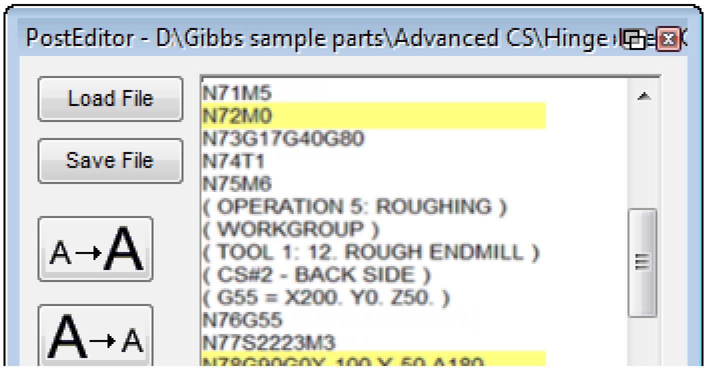
1. Create this Holes process with tool #4.



All of the operations to machine both the top and bottom of the part have been completed.



When using work fixture offsets to machine different sides of a part, a B-Style Advanced CS Processor should be used to post the file. The operations that machine the top side of the part will all be done in G54. Then there will be a program stop in the program so the operator can flip the part. The back side operations will be done in G55 so the origin shift for G55 must be entered for the work fixture offset correctly at the machine.

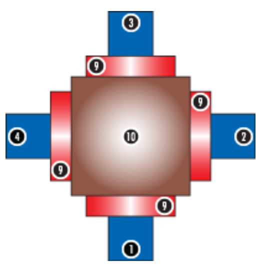


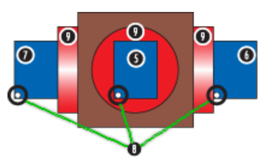
The highlighted rows show the Program Stop and the rotation.

TOMBSTONE MACHINING

Tombstone Machining

There are several ways to accomplish tombstone machining with the system. All of them require the Advanced CS option and a CNC machine capable of handling B-axis numbers in the WFO. The post processing is the same for all methods. The following three sections go over the different methods. The pictures shown below show our perspective of the tombstone and parts. For the dimensional data for the part print, see “Tombstone Machining” on page 109.





1. First Part
2. Second part
3. Third part
4. Fourth part
5. Front view of first part
6. Left side of second part
7. Right side of fourth part
8. Origins
9. Clamps
10. Tombstone

Top view Front view

Top and Front views of a Tombstone Setup

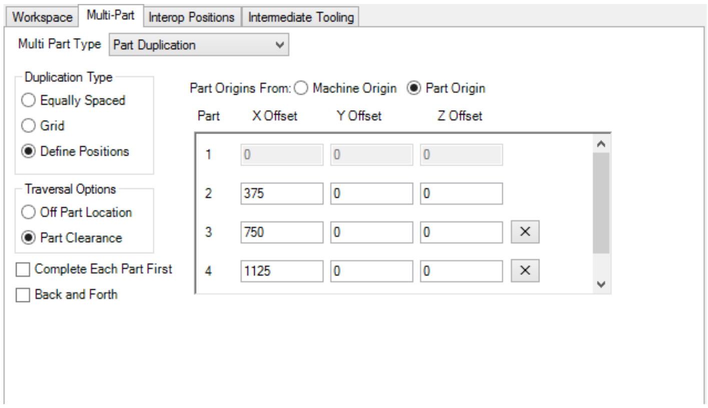
Method 1: Per the Machine

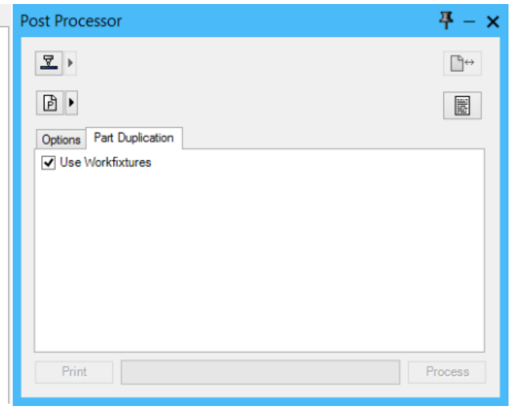
In this example we will demonstrate what will actually happen at the machine. The initial stock size encompasses the tombstone and the first, second, and fourth parts. For this example, we are looking at the “front” view, as shown above.

We will create the geometry and operations necessary to machine everything available from the “front” view; the front side of the 1st part, the right side of the 4th part and the left side of the 2nd part.

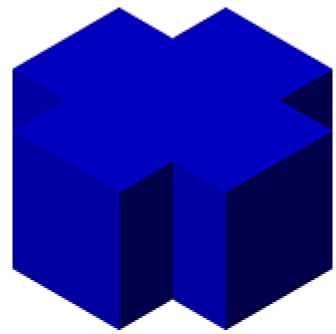
Then the post processor will be used to duplicate this information for the other three sides of the tombstone. This method has the benefit of allowing us to see the tool move from the side of one part to the face of another so that we can check for tool interference.

For the sample part that accompanies this tutorial, see the sample file Tombstone Method 1.vnc.



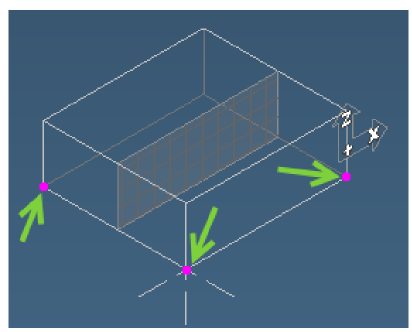


Part and CS Setup



1. Set up a 4-axis horizontal part with the stock size large enough to encompass the first part, second part, fourth part, and the tombstone including the clamps.

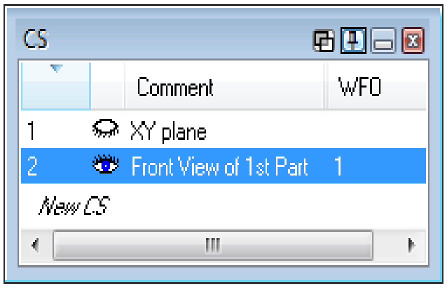
We will create three coordinate systems, one for each side of the part. All of these coordinate systems will be set in the XY plane, but will have different origins. The advantage to setting three separate planes is that the information entered in the machining process dialogs such as the clearance positions and cut depths are relative to the origin and coordinate system in which you are working. The images in this tutorial show a part with custom stock that encompasses the tombstone and all four parts.

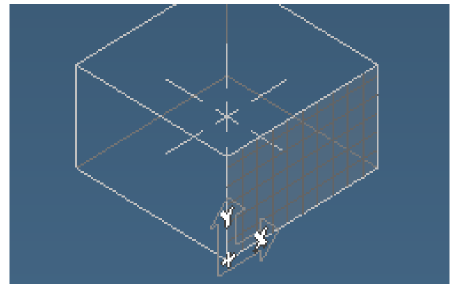


2. In the standard XY plane create origin points for each of the sides.

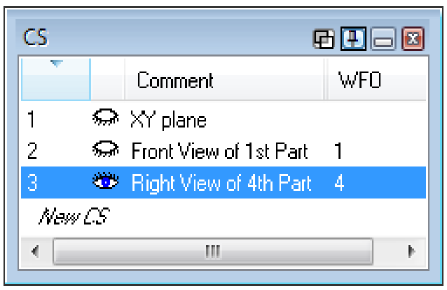
These origin points can lie anywhere in the plane. For the part that we have included for the first example we chose the bottom left hand corner of each side to act as the origin.

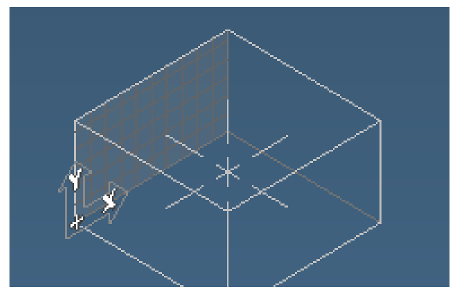
3. Open the CS dialog and dock it on a convenient side of the screen, as you will be using this extensively. (Click the dock symbol on the dialog and drag the CS titlebar until the arrow cursor is situated over the area you wish to dock to.)
4. Create a new CS. Change the origin to the origin point you created for the 1st part. Label this CS `Front view of 1st part`.



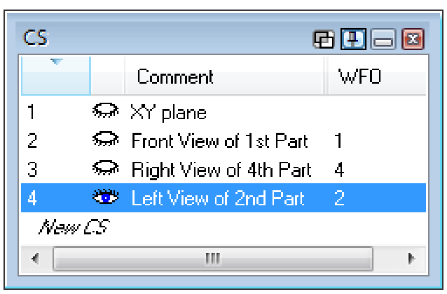


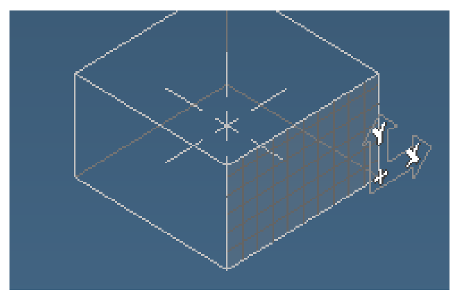
5. Create a new CS (CS3). Change the origin to the origin point you created for the 4th part. Label this CS `Right view of 4th part`.





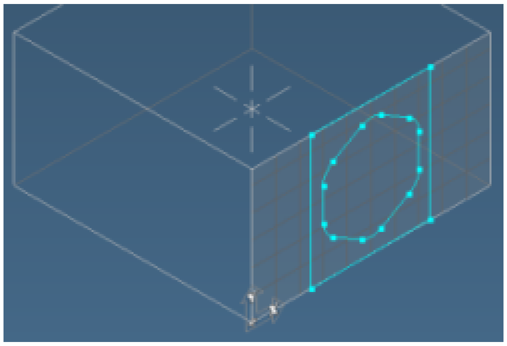
6. Create a new CS (CS4). Change the origin to the origin point you created for the 2nd part. Label this CS `Left view of 2nd part`.

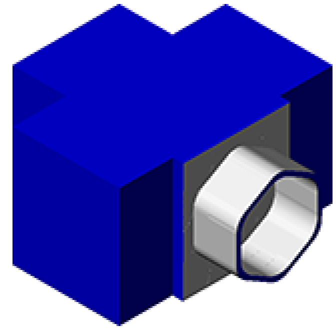




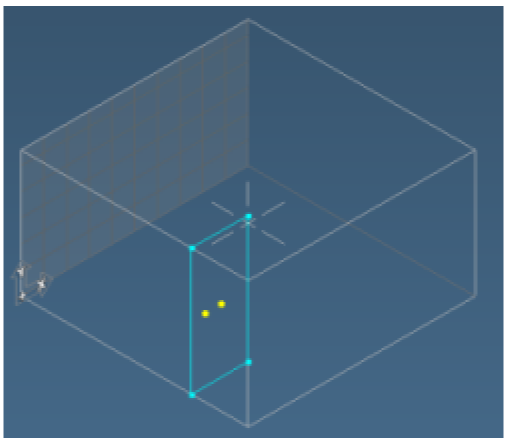
Machining

1. Switch to CS2. Create a new workgroup. Label this workgroup `Front of 1st part`. In order to better visualize what's going on, create an outline for the stock size of the front of the part. Create the geometry and operations for the machining of this side.

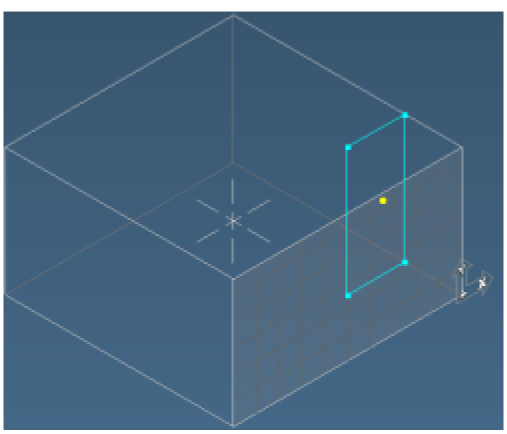


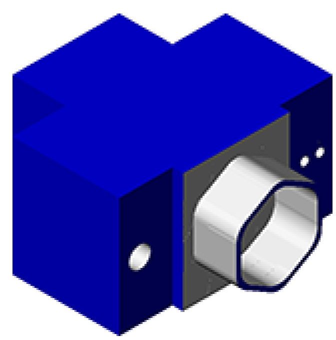


2. Switch to CS4. Create a new workgroup. Label this workgroup `Left side of 2nd part`. Create the stock outline for the left side. Create the geometry and operations for the machining of the left side. In the sample file there are two holes on the left side.



3. Switch to CS3. Create a new workgroup and label this workgroup `Right side of 4th part`. Create the stock outline for the right side. Create the geometry and operations for the machining of the right side. In the sample file there is one drilled hole on the right side.





The rendered image will not look correct unless extra operations are created to clear off the extra stock between the parts. This can be done by creating contour operations with a large tool to wipe off the excess on each side. These operations should be thrown away before post processing. In our example we have cleared off the excess stock. This is not necessary, but does make the rendering process more visually representative.

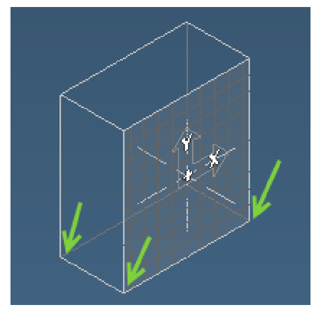
Method 2

In this example we will create one complete part, as if we were simply machining a single unit. This allows us to verify that the toolpaths will generate the correct part. It is especially useful when checking to make sure cuts made in one side intersect cuts made on another where they are supposed to.

When we're finished, we can do one of two things. We can either post process the file as is and strip out the extra B moves from a C or D style processor (there will be three of them,) or we can create new coordinate systems like in example 1 and "unfold" the part so it looks like it would on the tombstone. Unfolding the part is more work inside of the system, but it does help us see the part as it would be machined.

Part and CS Setup

1. Set up a 4-axis horizontal part. The stock dimensions should be that of a single part.



2. We will create three coordinate systems, one for each side of the part. The first CS will be for the front side of the part and will lie in the XY plane. The second CS will be for the right side of the part and will lie in the -YZ plane. The third CS will be for the left side of the part and will lie in the YZ plane.

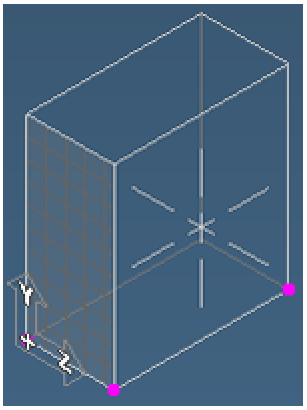
3. Again, in the standard XY plane, create origin points for each of the sides.

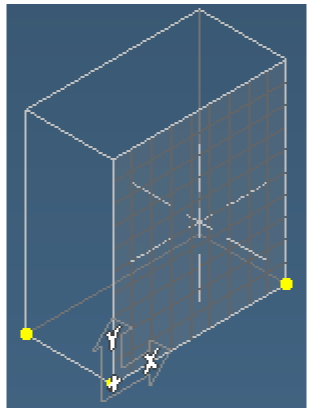
   These origin points can lie anywhere in the plane. For the part that we have included as Example 2, we chose the bottom left hand corner of each side to act as the origin.

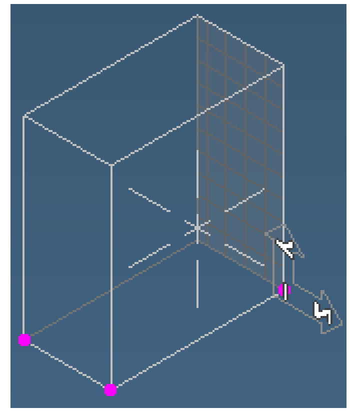
4. Create a new coordinate system (CS2). Change the origin to the origin point you created for the front side of the part. Label this CS `Front side`.

5. Create a new CS (CS3). Set the plane to the YZ plane. Change the origin to the origin point you created for the right side of the part. Label this CS `Right side`.

6. Create a new CS (CS4) Toggle the depth. Change the origin to the origin point you created for the left side of the part. Set the plane to the YZ plane. Label this CS `Left side`.

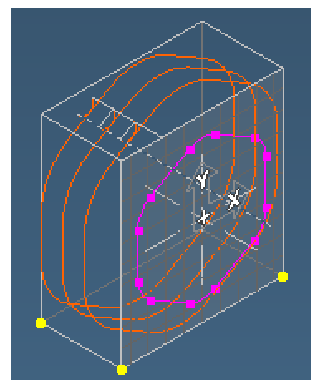




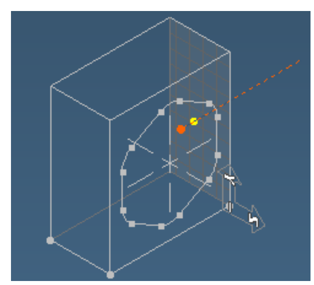


Left side   Front side   Right side

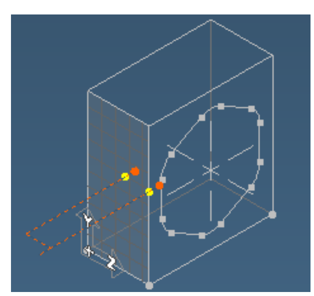
Machining



1. Switch to CS2. Create a new workgroup and label the workgroup `Front side`. Create the geometry and operations for the machining of the front side. In the sample file, Tombstone Method 2.vnc, there are 2 operations on the front side of the part.

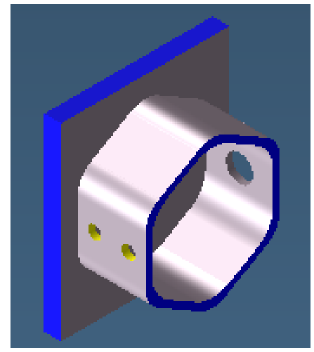


2. Switch to CS4. Create a new workgroup. Label this workgroup `Right side`. Create the geometry and operations for the machining of the right side. In the sample file there is one drilled hole on the right side.



3. Switch to CS3. Create a new workgroup. Label this workgroup `Left side`. Create the geometry and operations for the machining of the left side. In the sample file there are two holes drilled on the left side.

4. If you render the part at this point you will be able to see what one finished part will look like.

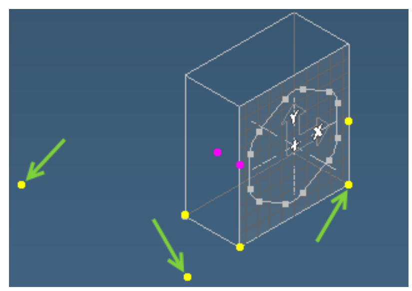


If you are using a B or C style post processor, you can post process the part now. If you process at this point you will need to remove the B axis moves and change the numbers at the control for the origins of the WFOs. If you are using a D style post processor, you must “unfold” the part before post processing.

Unfolding a Part

When you “unfold” a single part, it will work the same as method 1. To “unfold” a single part, we need to change the three coordinate systems so that they match the ones we created in Example 1. The sides will automatically unfold out and move into position. We will also change the stock size to encompass parts one, two, and four, as well as the tombstone.

1. Switch to CS1, the XY plane. Create origin points for the three side. These origin points will be in the same locations as the origin points in Example 1, Front view.



2. Switch to CS2, the Front side. Change the origin to the new origin point you created for the Front side. When you switch the origin of the plane, the operations that you created in that coordinate system should shift accordingly so you do not need to adjust or reprocess the operations.
3. Switch to CS3, the Right side. Change the origin to the new origin point you created for the Right side. Note that the origin for the plane containing the right side is actually on the far left side when it is unfolded because in this setup we are machining the right side of the fourth part which is attached to the tombstone on the left. (For more information, see “Top and Front views of a Tombstone Setup” on page 82.) Also, you need to set this CS to the XY plane.
4. Switch to CS4, the Left side. Change the origin to the new origin point you created for the Left side. Again, the origin for the left side appears on the right side in this set up. Also, you need to set this CS to the XY plane.

Method 3

This example is designed to reduce the number of WFOs needed to machine the part. Instead of creating different coordinate systems, we will simply use the standard XY plane. You only need one WG and one CS. All geometry is created and machined in the standard XY plane. Basically, this is how tombstone milling would be accomplished without using the Advanced CS module. Using this method the depth values need to be calculated and entered in the process dialogs. The rendered image will have the same problems as Example 1. To solve this you will need to create temporary operations to remove the excess stock. This method is useful if your machine can only handle a small number of WFOs.

Post Processing

Using any of the methods outlined above, once you have created the part, post process the file using work fixture offsets and 4 parts. Advanced CS post processors come in three styles.

B Style:
WFOs used for origins and indexing for single and multiple parts.

C Style:
WFOs used for origins for single parts. WFOs used for origins and indexing between parts for multiple parts.

D Style:
WFOs only used for indexing between multiple parts.

If your CNC machine does not have the number of WFOs required to machine the part, B and C style processors will revert to D style.

The indexing must be handled by the work fixtures. The first three work fixtures will be at B0, the next three at B90, the next three at B180, etc. If there are partial rotations for the parts, they will be handled by creating additional coordinate systems and will result in more than three coordinate systems per side. The actual rotation to get to these other sides will show up as “BXX” commands in the code for the C and D style processors. Since each side will have the correct rotation for the part in the WFO, the partial rotations will work fine. The B style post will output a comment containing the partial rotation amount. For the parts on the second, third, and fourth sides of the tombstone, the 90° index increment will need to be added to the partial rotation amount.

CREATION AND MACHINING CONCEPTS

This chapter contains short, how-to tutorials. Each tutorial is focused on a topic, such as making a bottle mold, working with multiple part origins or driving a tool along the Z axis.

- Multiple Part Origins
- Bottle Mold
- Driving the Tool In Z
- Rotary Milling

Multiple Part Origins

Using Multiple CS's for Geometry Creation

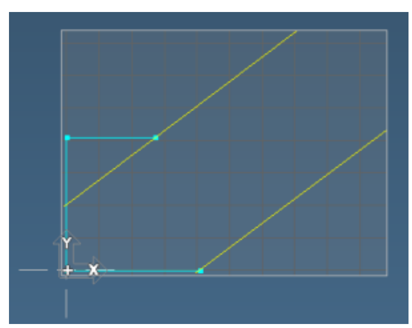
In this exercise, we will use multiple part origins to simplify the creation of geometry. The different part origins will not be used with work fixture offsets as they were shown in previous exercises, see "Family of Parts" on page 61, or "Top and Bottom Machining" on page 71. Rather, it is the way the part is dimensioned that makes using different part origins the best way to create the part geometry. This exercise uses a standard Mill post processor and a 3-axis MDD. For the part print for this exercise, see "Multiple Origins" on page 111.

Part Creation

1. Create a new metric part using a 3 Axis Mill MDD with a stock size of X=-5, 300; Y= -5, 225; Z= -25, 0.

   First, we will create the outside contour of the shape. All of the lines that compose the outside contour, except for one, are easily created in CS1.

2. Create the geometry for the outside contour that is dimensioned from the 0, 0 origin on the part print.

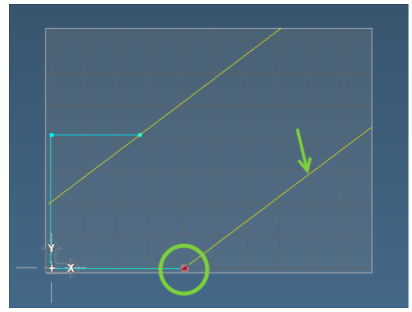


Second CS

The line that connects the angled lines to complete the outside contour is dimensioned from a different origin on the print. While we could figure out the dimensioning for this line, it would take a bit of work. Instead, we will create another coordinate system with a new origin.

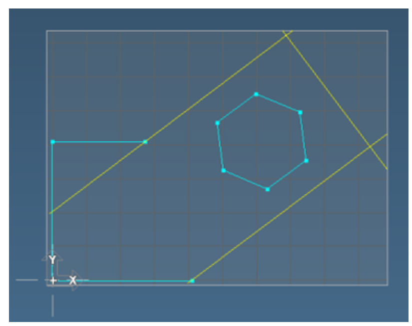
1. Create a new CS and select the point circled as the origin.

   The new CS is an exact duplicate of the standard XY plane; the only thing that has changed is the origin. To further modify the new CS, we will set the horizontal axis to one of the angled lines. This will not change the planar location of the coordinate system, only the CS grid orientation.



2. Align the horizontal axis to the line shown with the arrow above.

   Now, the angled line that completes the outside contour is very simple to create. You can simply define the line as a vertical line with an H value of 200mm.



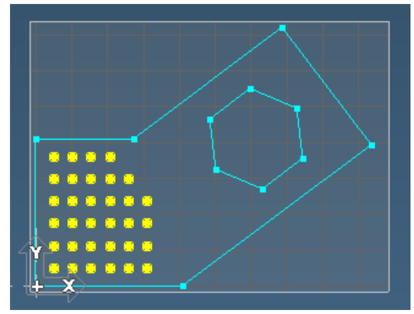
3. Create the line to finish the outside contour and the hexagon while working in CS2.

Completing the Geometry

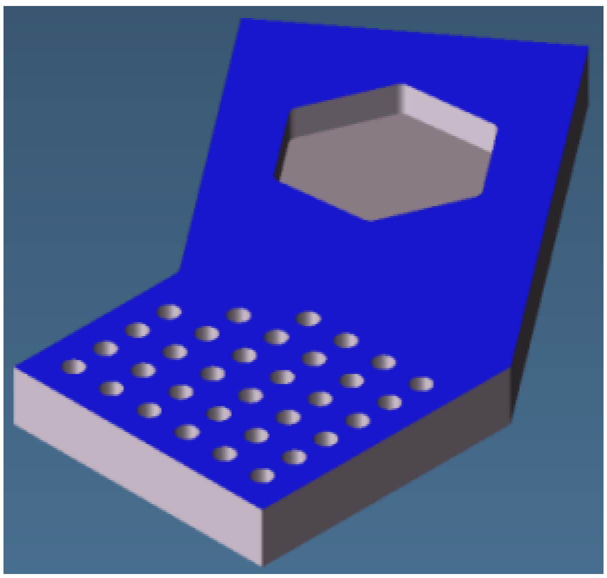
1. Switch to CS1 and select the hexagon and line we just created in CS2.
2. Choose Modify > Change CS (XYZ).

   All of the part geometry should now be either yellow or blue in CS1 indicating that it is based on the current CS.

3. Connect the outside contour and create the bolt hole pattern to complete the part geometry.



The part can be machined just as it would be if you were working with the Production Mill Module. All operations should have the Machining CS in the process dialogs set to CS1: XY Plane.



Bottle Mold

In this exercise we will make a bottle mold. This exercise uses a pre-existing part file that contains the bottle profile and a tool.

About Bottle Molds

Bottle molds are created in the system using the 2 1/2 axis surfacing capabilities. More specifically, bottle molds are created using the swept shape option for machining contours. Swept shapes are created by designating a base curve, which is the cut shape, that will be machined based on a drive curve. The drive curve designates the shape of the wall that will be swept around the base curve to create the surface.

For bottle molds, the base curve is an arc and the drive curve is the contour that represents the shape of the bottle. Normally, the base curve is defined in either the XZ or YZ planes and the drive curve is defined in the standard XY plane. All machining will be done from CS1, which is the standard XY plane.

The general rule when creating bottlemolds is that the depth axis of the base curve must align with the vertical axis of the drive curve (the bottle profile). When the Machining CS and the drive curve plane are perpendicular, the system will output G17, G18 and G19 codes to provide for the machining of the arc rather than breaking it into line segments.

Creating the Bottle Mold

For this part, we will create the base curve, which is a 13mm arc in the XZ plane. The drive curve, which is the shape of the bottle in the XY plane, is already created to save time. We will need to properly position the drive curve to complete the part. If you need to recreate the profile shape for this exercise, see “Family of Parts” on page 108.

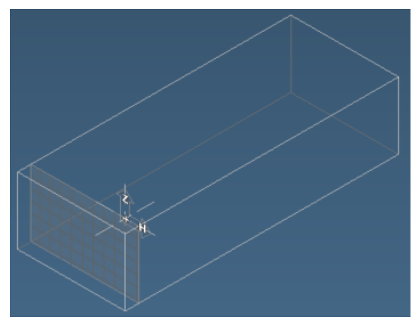
1. Open the file Bottle Mold.vnc, which is located in the Part Files > Advanced CS folder.

   This part is set up to use a 3-Axis Vertical Mill. The Clearance Plane value of 15mm is plenty of clearance over our X:-10/+193, Y: -10/+70, Z: -50/0mm part.

Creating the Base Curve

First, we will create the base curve geometry, which is a 13mm arc in the YZ plane.

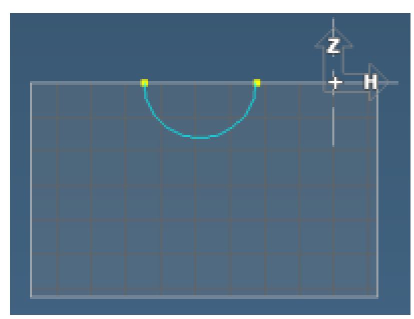
1. Create a new CS.
2. Modify the CS to the YZ plane.
3. Click the Toggle Depth button.



In this coordinate system we will create the base curve geometry, which will be an arc.

4. Create a 13mm circle in the YZ plane around the centerpoint H -31, Z0, D0.

   We need to create terminator points to create the semi-circle (arc) that will act as our drive curve.

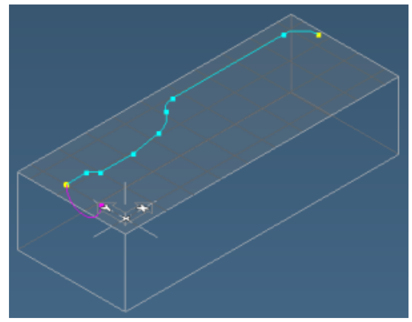


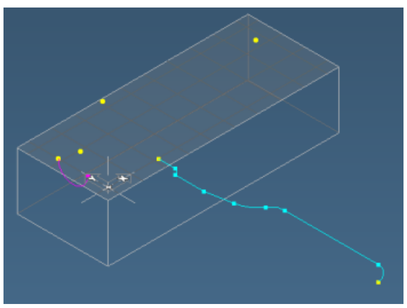
Now, we will create the drive curve geometry which is the contour of the bottle. There must be terminators at both ends of the drive curve.

Correctly positioning the drive curve is very important in order to generate the desired toolpath. The general rule is that the vertical axis of the drive curve is going to align with the depth axis of the base curve.

In this exercise, we will create the drive curve geometry in CS1, which is the standard XY plane. Once the geometry is created, we will 2D Rotate the contour 90° so that the vertical axis of the drive curve will correctly align with the base curve depth axis.

5. Change to CS1 and create the drive curve geometry. Refer to the part print for the dimensions.



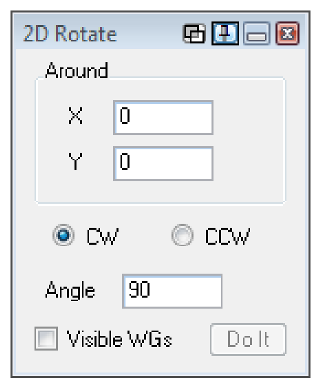


The drive curve geometry does not need to be created in position because we will be rotating it. However, it is drawn in the screenshot on the left in position for visualization purposes.

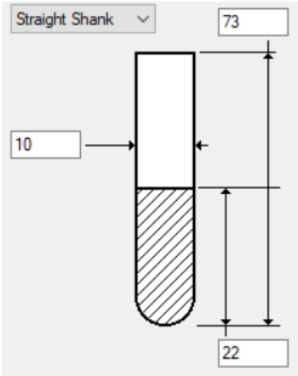
Now, we need to rotate the drive curve 90° so that it is oriented correctly.

6. Make sure that you are working in CS1. Select the drive curve shape. Choose Modify > 2D Rotate and enter the information shown.

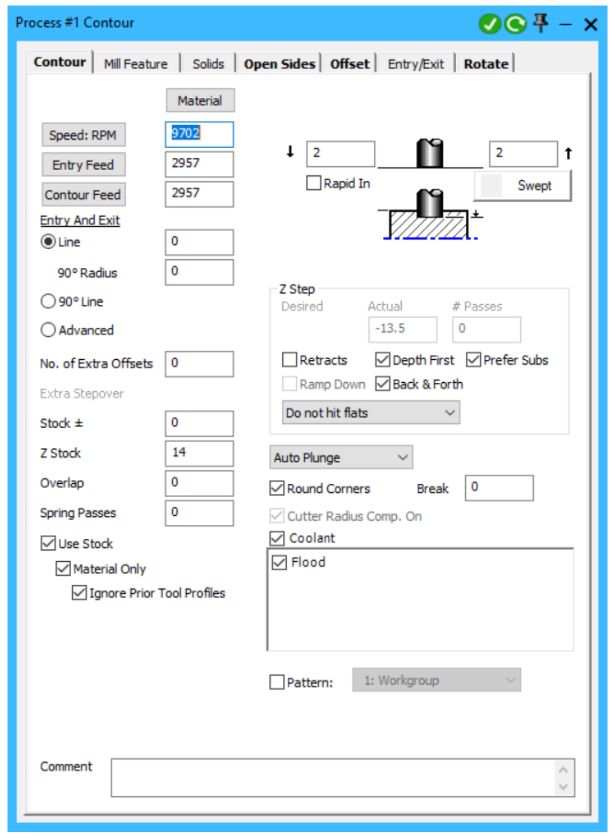
   The orientation of the drive curve is what is important, not its exact location in the coordinate system. As long as the shape is rotated 90°, the point around which it is rotated is not important. The location of the bottle mold toolpath will be determined by the base curve, which is in position.



Next, we will create contouring processes which will generate the bottle mold toolpath.

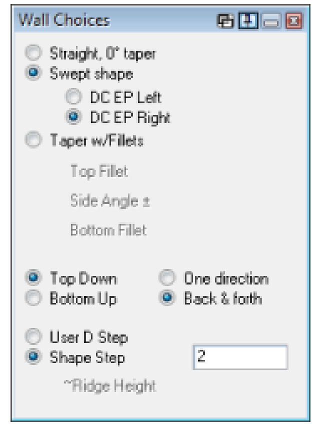


7. Create a 10mm Ball Endmill.

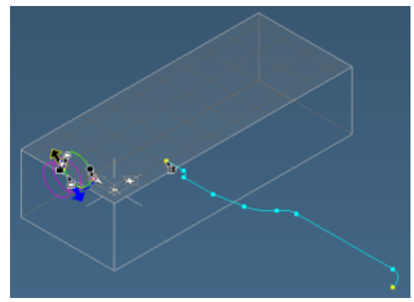


8. Create this Contour process with the ball endmill.

Since our flute length is smaller than our final depth of cut we will need to do this contour in several Z passes. Z stock will create the contour toolpath and translate the cut in Z the distance specified. We will take two passes with Z stock applied before taking the final cut. The only information we really need to be concerned with is entered in the Wall Choices dialog.

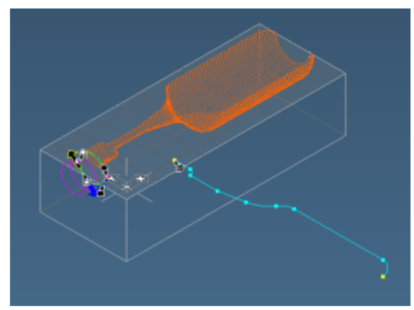


9. Click the Swept button and enter the information shown.



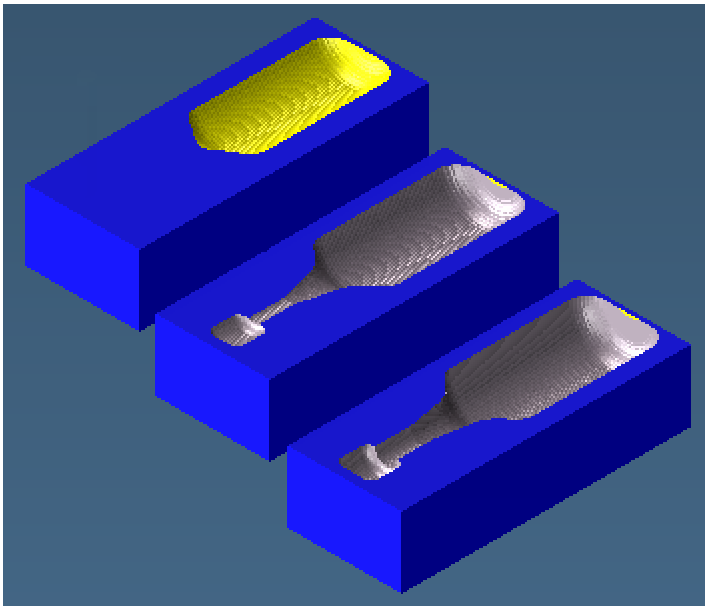
10. Select the arc as the base curve. Set the direction to contour into the arc.

    The machining markers that appear for contouring operations will come up on the screen. The D-marker, which designates the drive curve for swept surfaces, will also appear.



11. Click the Do it button.

12. Using the same process, create 2 more contour ops with Z Stock at 7 and then 0.



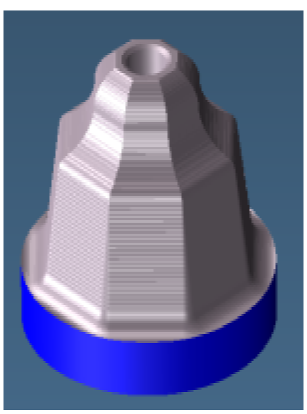
Driving the Tool In Z

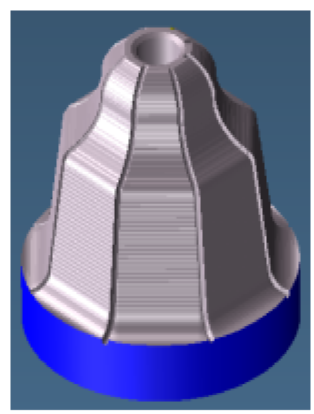
About the Part

In this exercise we will start with a mostly completed part. We will add the finishing touches by adding grooves along the edges of the part.

1. Open the part file Z Machining.vnc from the Part Files > Advanced CS folder.

   If you would like, now is a good time to run cut part rendering as it will help you familiarize yourself with the part.





Start	Finish

Setting Up the Part

1. From the XZ plane copy (`Ctrl+C`) the profile geometry (the magenta geometry).

2. Create a new WG called `Grooves` and a new CS called `CW 22.5`.

3. Paste the copied geometry into the Grooves WG.

   You may note that the geometry is still magenta. That is because the geometry is still assigned to the XZ plane. We will need to change that.

4. If you are not already in it, switch to the CS CW 22.5.

5. Assign the geometry to the current CS using Change CS (XYZ).

Modifying the Geometry

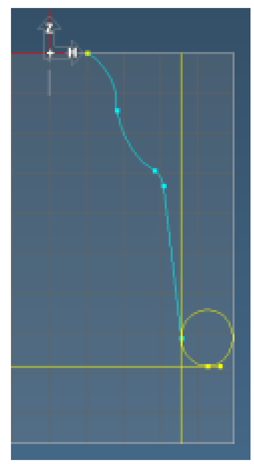
Profile Shape

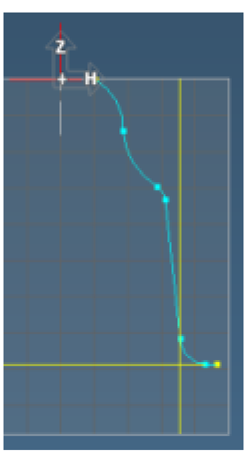
We used a 13mm BEm to cut the part, so we have a 6.5mm radius on the bottom of the part. We will be driving the tool onto the part and up the profile but the profile stops at the top of the fillet. Therefore, we need to create the geometry to replicate the fillet. This way the tool can start off the part and cut up the edge of the part.

1. Make a vertical line tangent to the right side of the profile.

2. Make a horizontal line 6.5mm below the profile.

3. Create a circle with a 6.5mm radius between the two lines. The circle will arc down and to the right from the profile, as shown in the image.





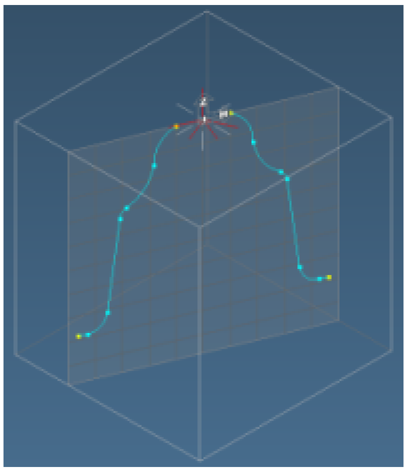
4. Connect the new arc to the profile geometry and terminate the line at the stock boundary (X=42.5mm).

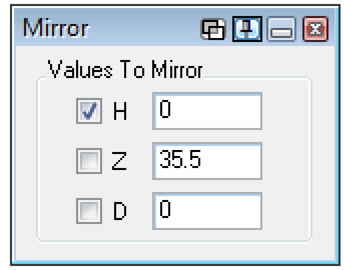
   You may delete the vertical line, but do not delete the horizontal line. In fact, make sure the horizontal line is connected to the arc – we will use it for driving the tool onto the part. The line does not need to be terminated.

Making the Grooves

To create the groove profiles we need to duplicate the geometry, and then duplicate and rotate the geometry and the CS into the proper positions.

1. Select the profile geometry. Duplicate and mirror the geometry about the X axis.





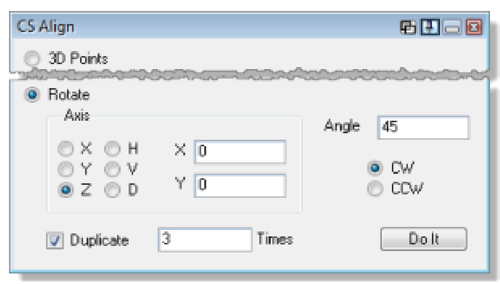
2. Deselect the geometry.

   You will now have identical geometry on both sides of H0. We now need to rotate the CS and geometry. Deselecting the geometry is important as we are going to be modifying the CS origins.

3. Open the CS Align dialog and rotate the CS clockwise about the Z axis by 22.5°.

   The geometry should move with the CS. If the geometry does not move, Undo the rotation and change the CS Preferences by Right-clicking the CS list title bar.

4. Rotate and Duplicate the CS 3 times by 45° around Z.



5. 2D Rotate and Duplicate the geometry 3 times by 45° around X0Y0 (from the XY plane).

   You will now have four pairs of profile geometry that align to four planes.

6. Assign each pair of geometry profiles to the planar CS.

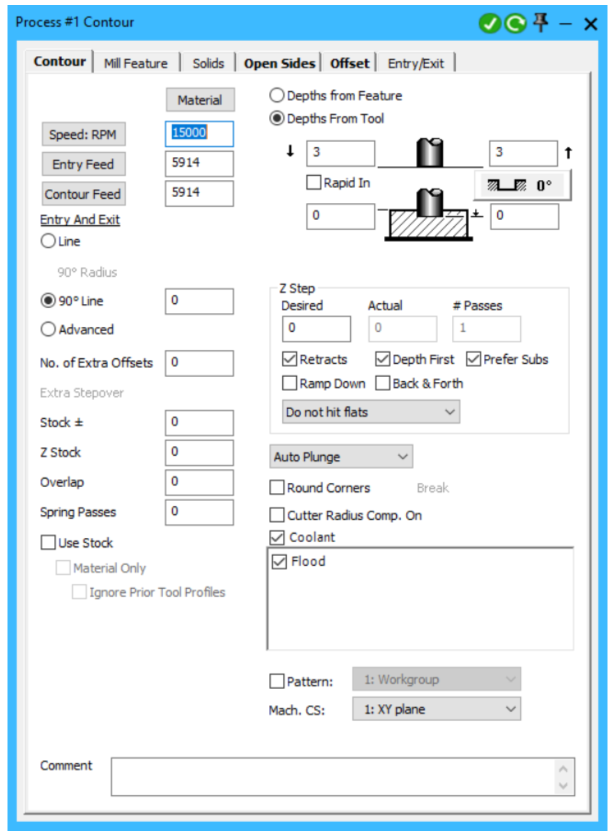
It is important to get the geometry into the proper CS at the proper depth because the machining depth is specified by the geometry's CS.

Machining the Grooves

Contour Operations

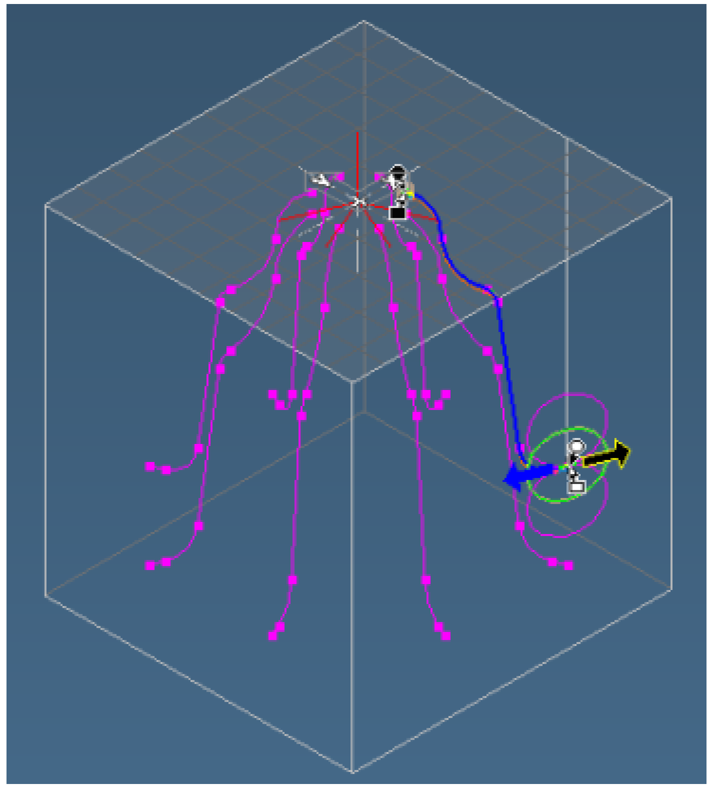
To machine the part we will need to select each profile shape and apply a contour process to it.

1. If you are not already in it, switch to XY plane.

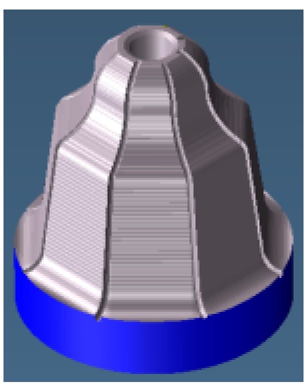


2. Select each profile shape geometry individually and apply the machining settings as shown to make an operation for each shape. You will need to repeat this seven times.

Be sure to deselect each operation tile before selecting the next profile geometry and making the next operation.



3. Render the operations.

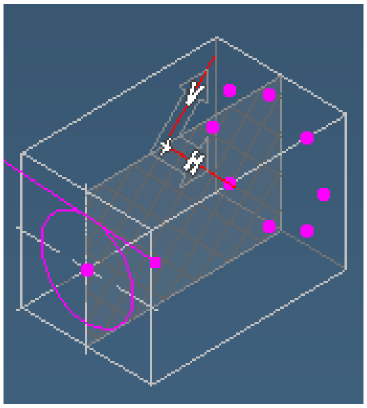


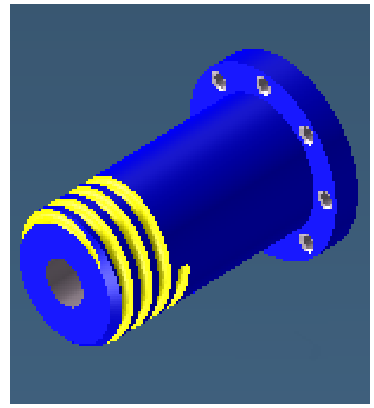
Rotary Milling

Rotary milling can be done in any CS whose origin is at the part origin and whose axis of rotation lies in the origin's planes. The rotary machining motion is always about the axis with 360° of rotation. Rotary milling may not be done around an axis perpendicular to the part's rotary axis, which means you may not do B-axis rotations or rotations about the 5th axis.

The Rotary Part

We will now do a quick exercise that goes over some of the ways coordinate systems may be used with rotary machining. We will start with an existing part with a rotary operation. We will make a new CS, define some text geometry and engrave the text shapes.





About the File

1. Open the part file Rotary Text.vnc.

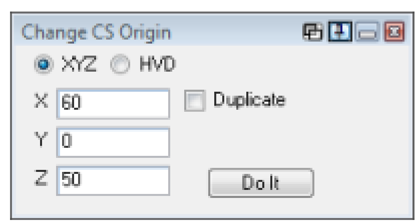
   This part is partially complete with pre-existing ops including 3 drilling operations and a rotary milling operation. This drilling operations require Advanced CS to make the -YZ plane and to machine from it. The rotary operation does not need Advanced CS. That is done in a Rotary Milling WG. One point is made, (in this case X-4, A-90, R35) a second point is made at the end of the spiral, including rotations (X46, A1350, R35) and a line is connected between the points. The angle is determined by multiplying the # of rotations by 360 plus the starting points angle, in this case (360x4)+(-90).

Making the Text

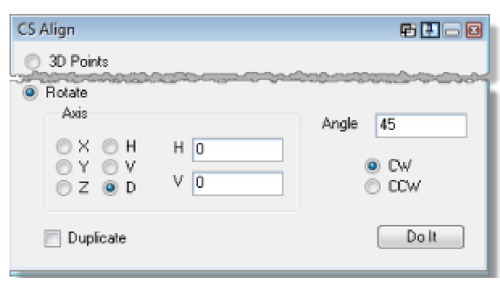
CS and Geometry

1. Make a new CS and set it to the XZ plane.

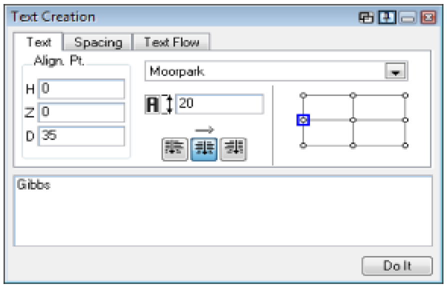
2. Move the CS origin as shown.

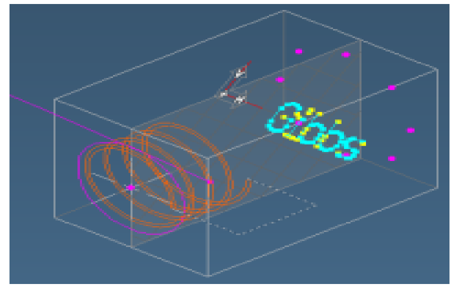


3. Rotate the CS about the depth axis by 45° clockwise as shown. Name the CS XZ mod.



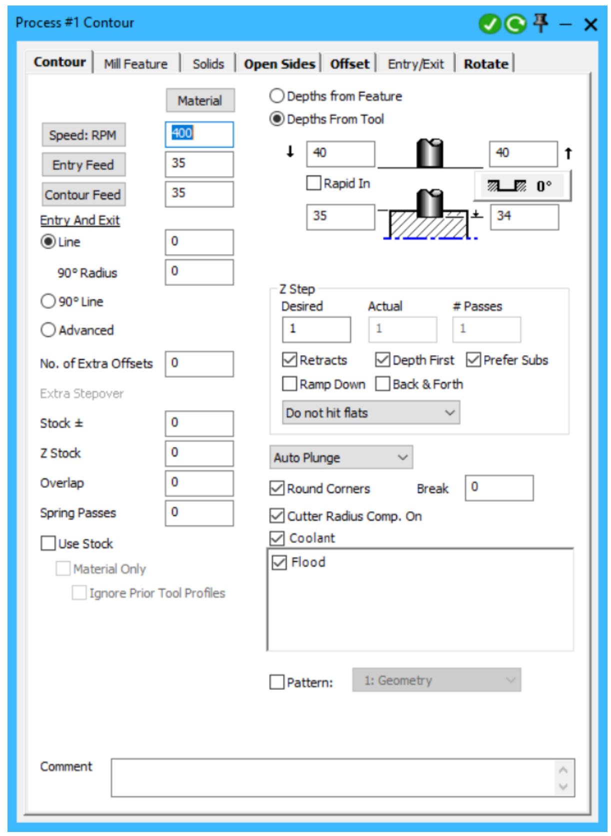
4. Create text geometry as shown below. Be sure that you are in WG1 as it is a rotary milling WG.



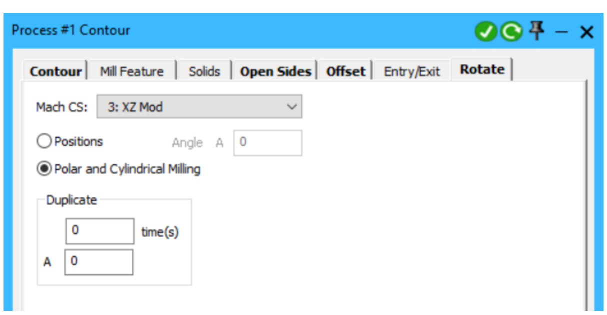


Machining the Part

Contour Process



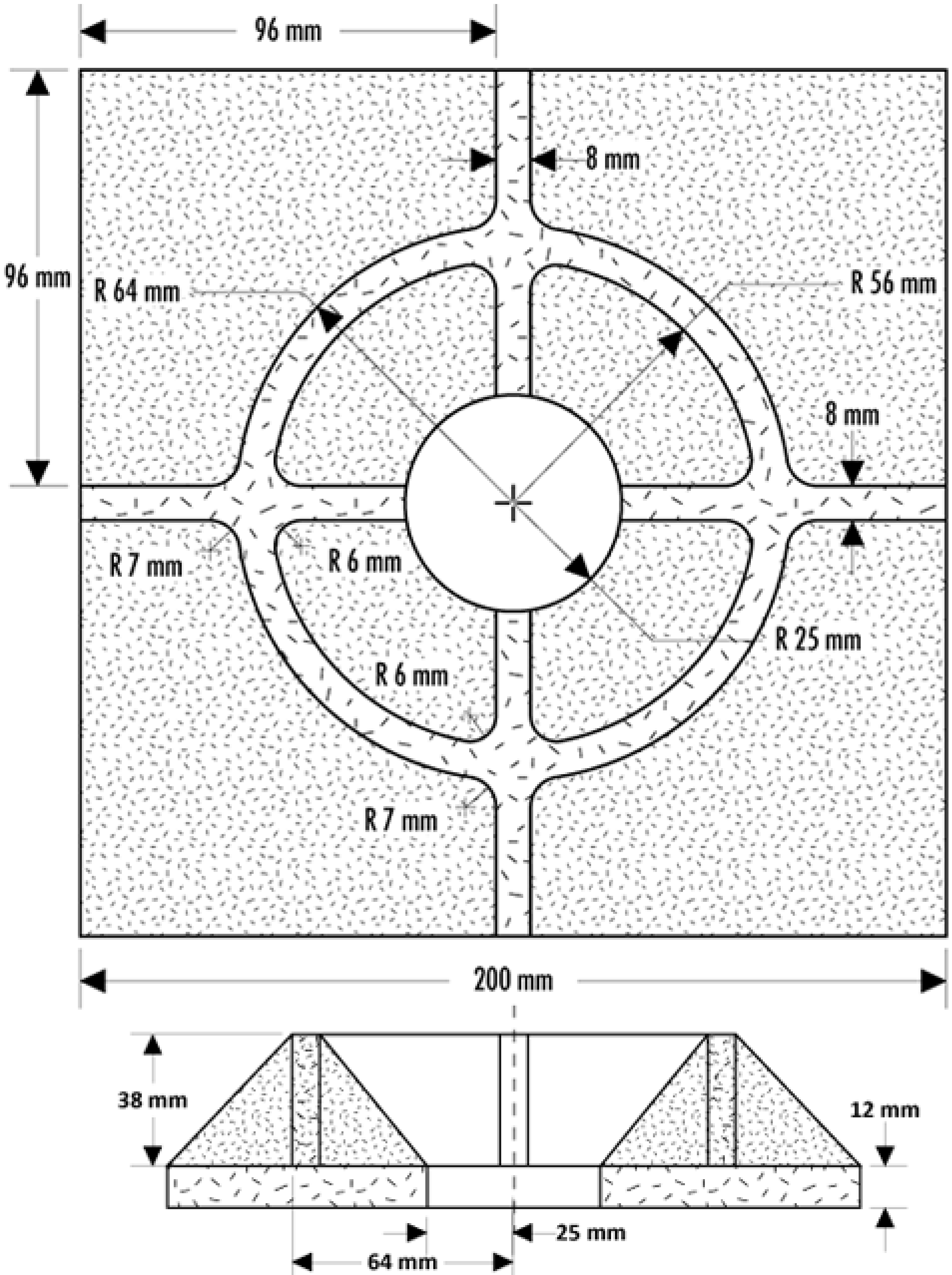
1. Using tool #5 (a 3.15mm center drill) create a Contour process to engrave the text using the parameters shown. Be sure to set the Rotate tab as shown below.

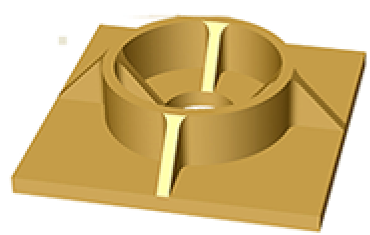


When rendered, the part will look like the following image.

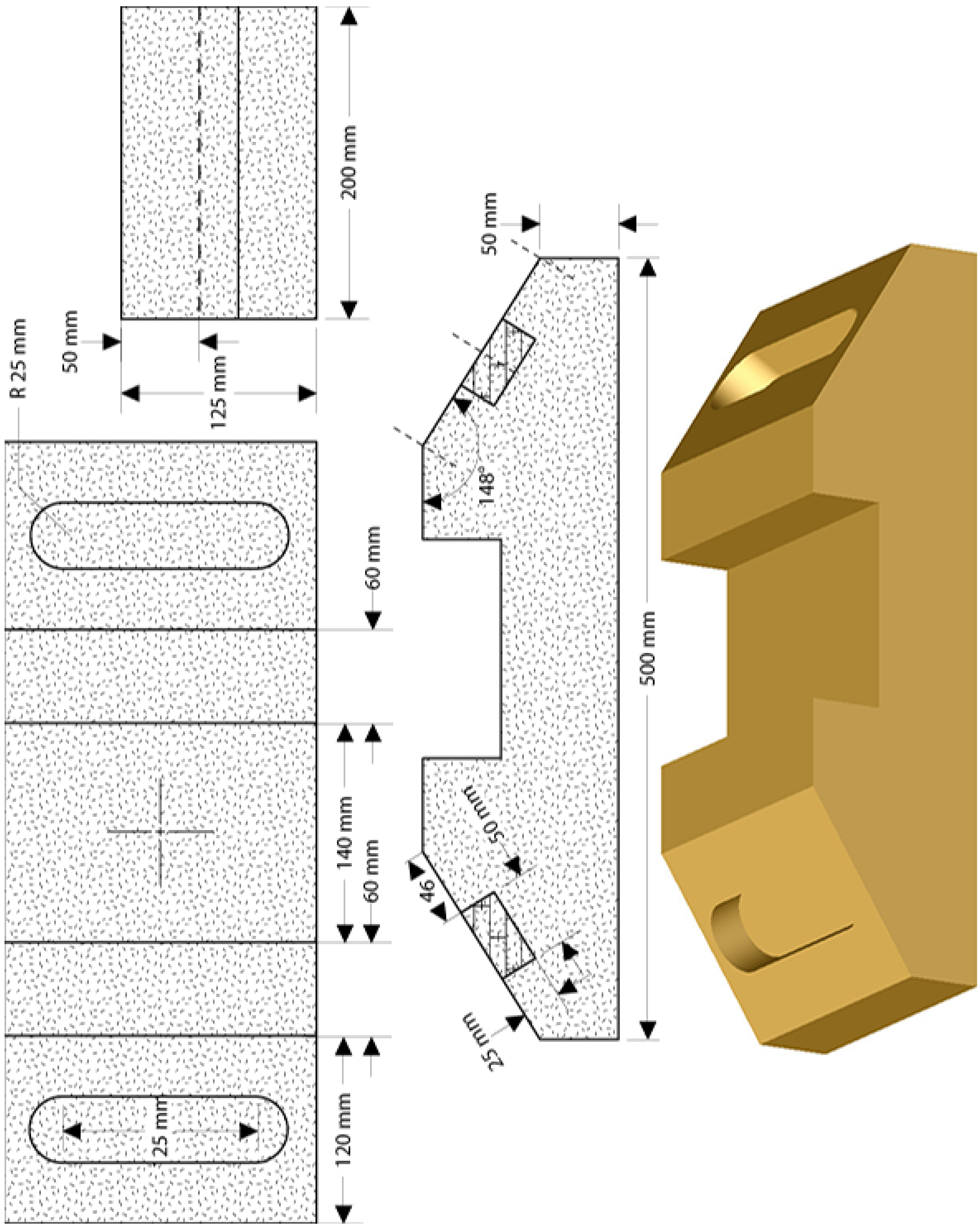
PART PRINTS

Angled Webs

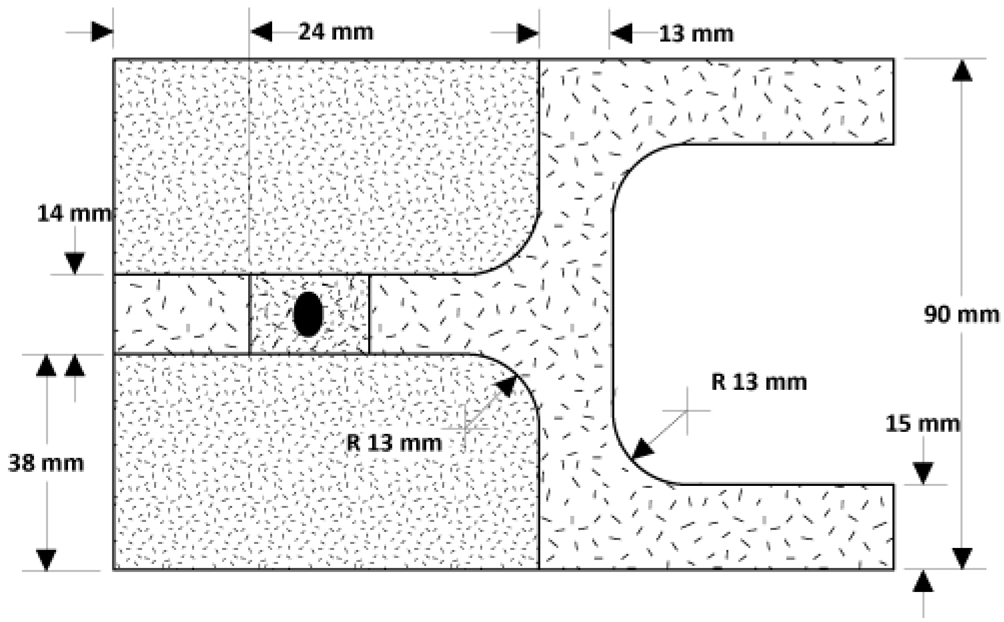


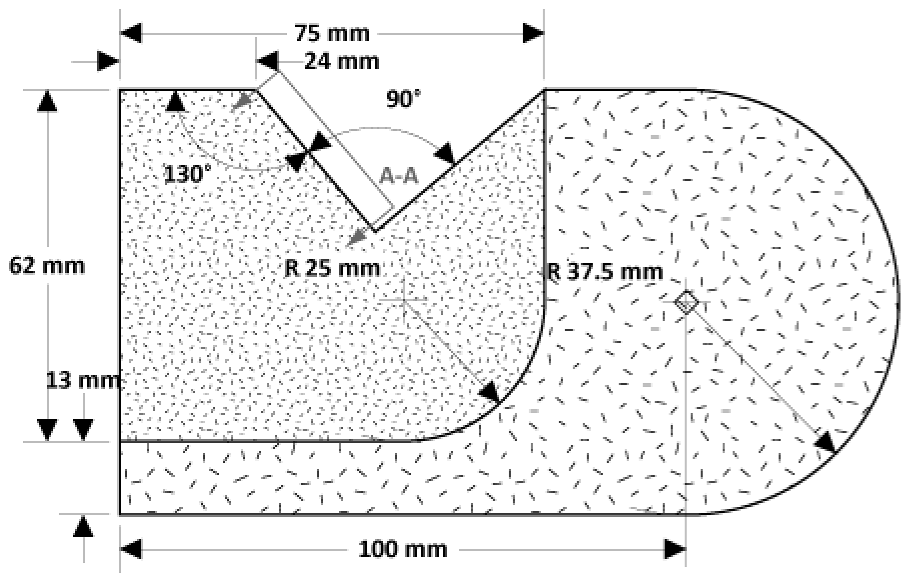


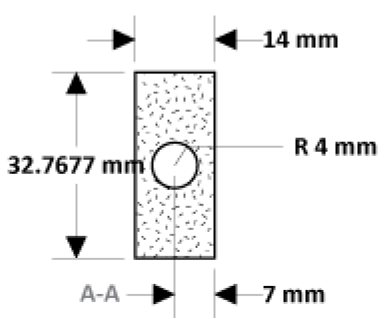
Slot Block

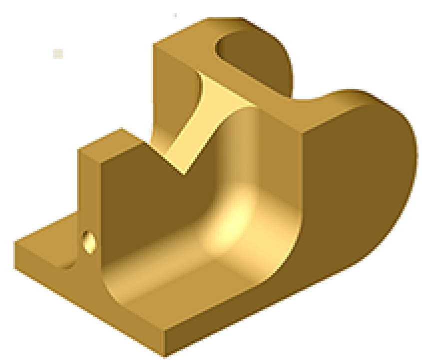

200 mm
50 mm
125 mm
R 25 mm
50 mm
148°
60 mm
500 mm
140 mm
60 mm
50 mm
46
25 mm
120 mm
25 mm

Hinge

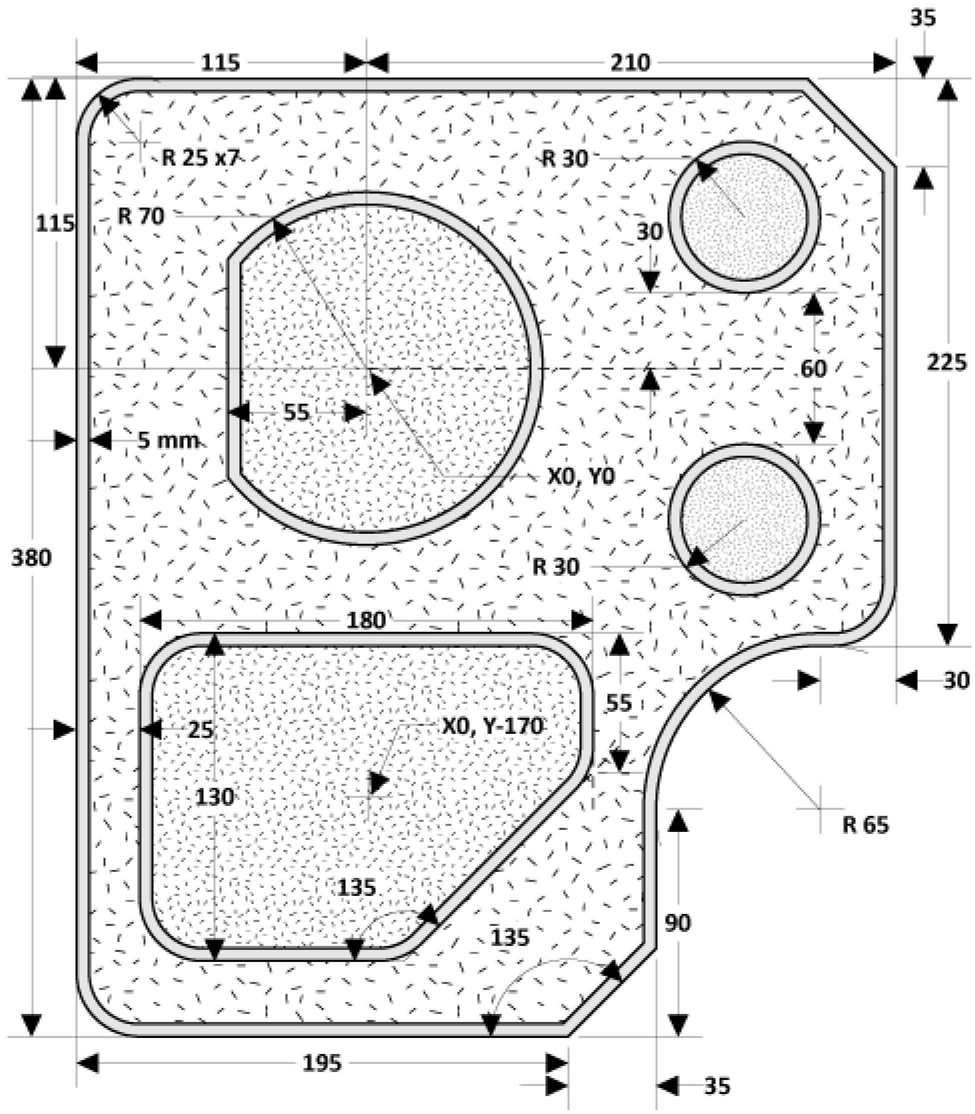

24 mm
13 mm
14 mm
90 mm
R 13 mm
R 13 mm
15 mm
38 mm

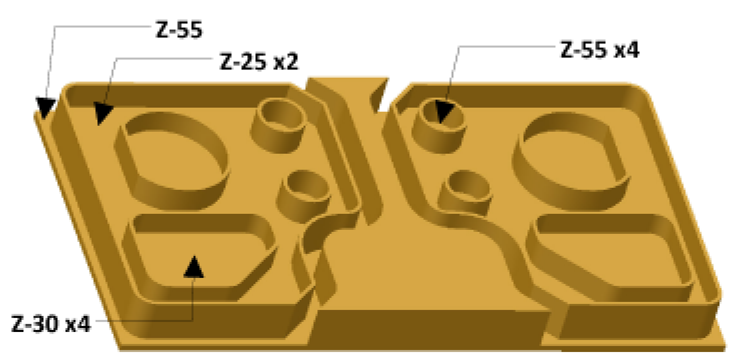

75 mm
24 mm
90°
130°
A-A
62 mm
R 25 mm
R 37.5 mm
13 mm
100 mm


14 mm
R 4 mm
32.7677 mm
A-A
7 mm

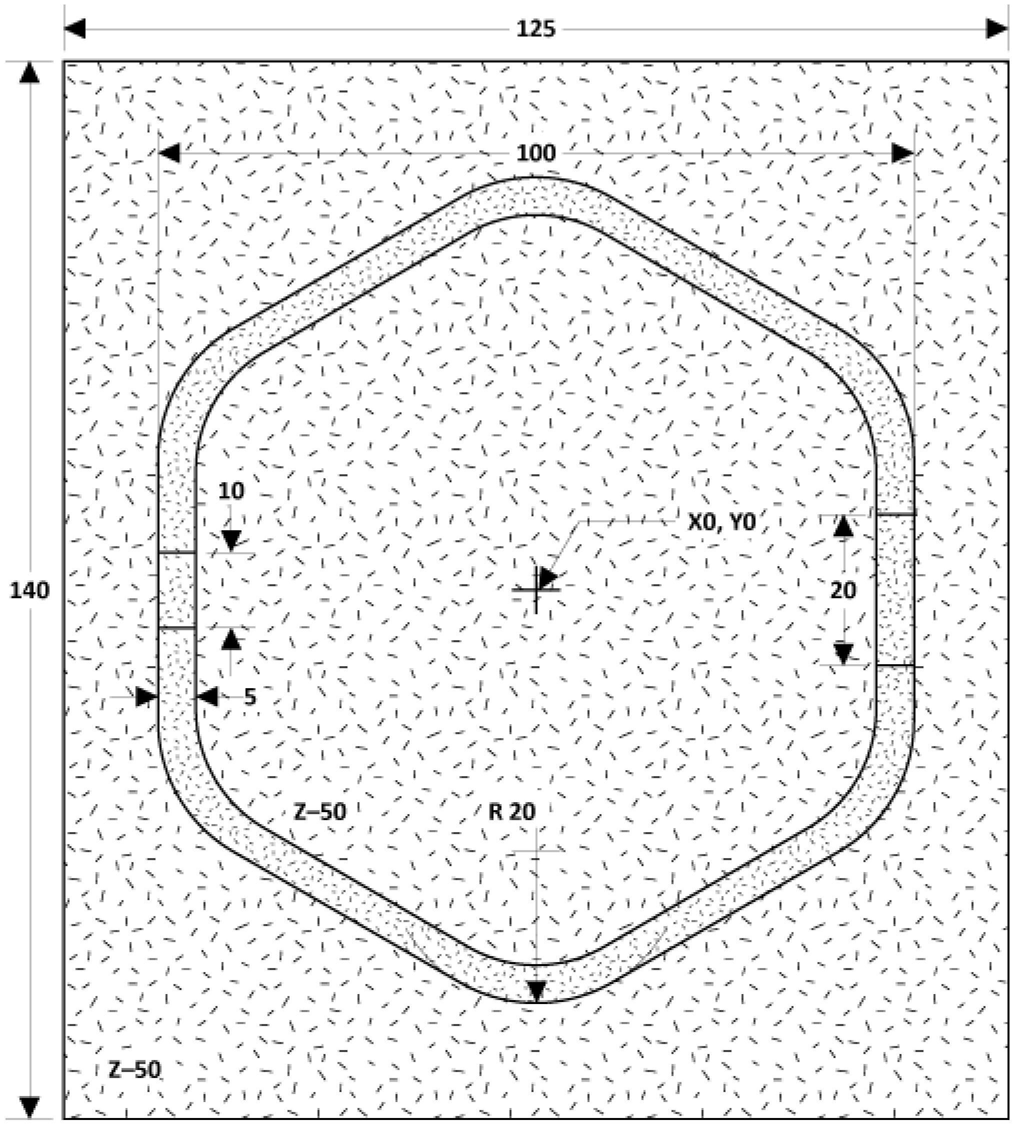


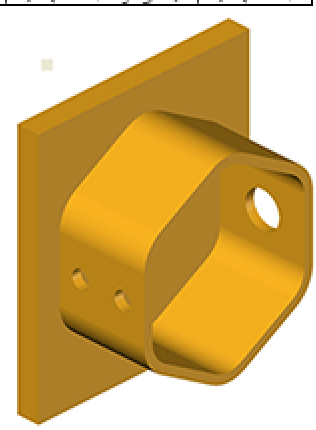
Family of Parts



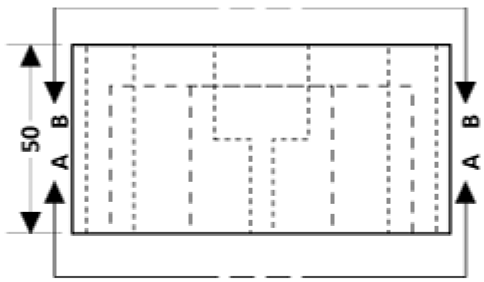


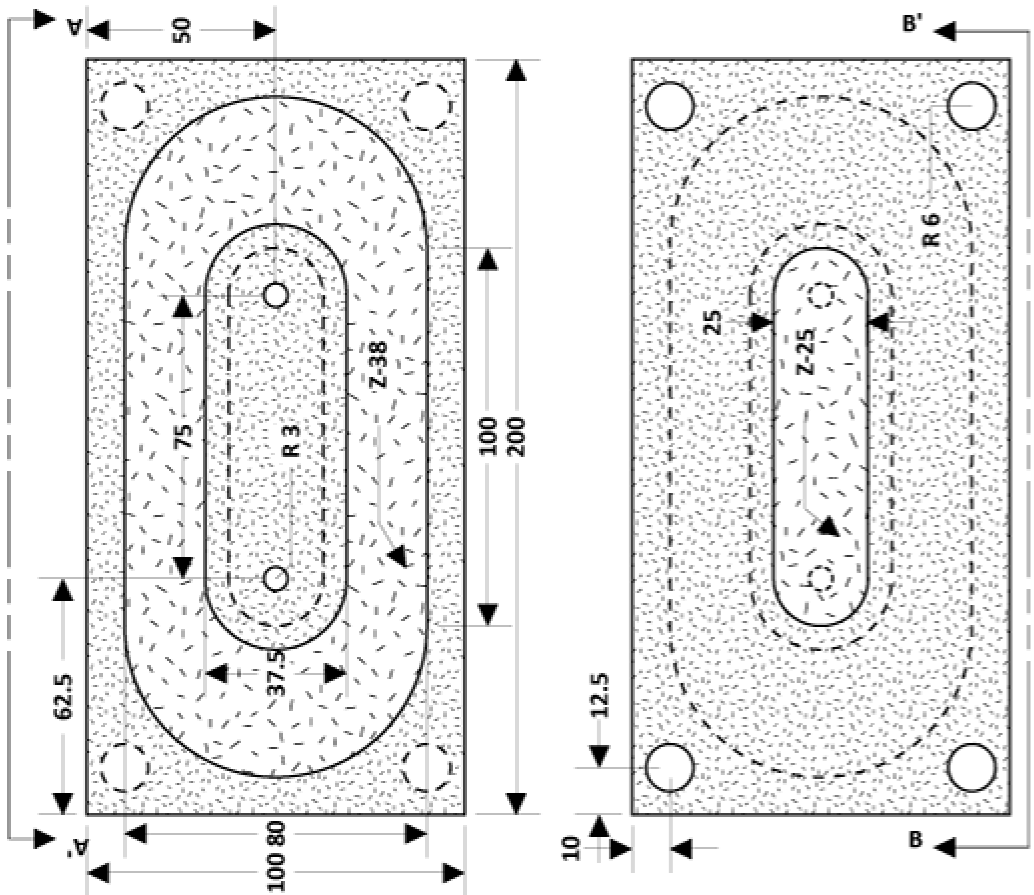
Tombstone Machining



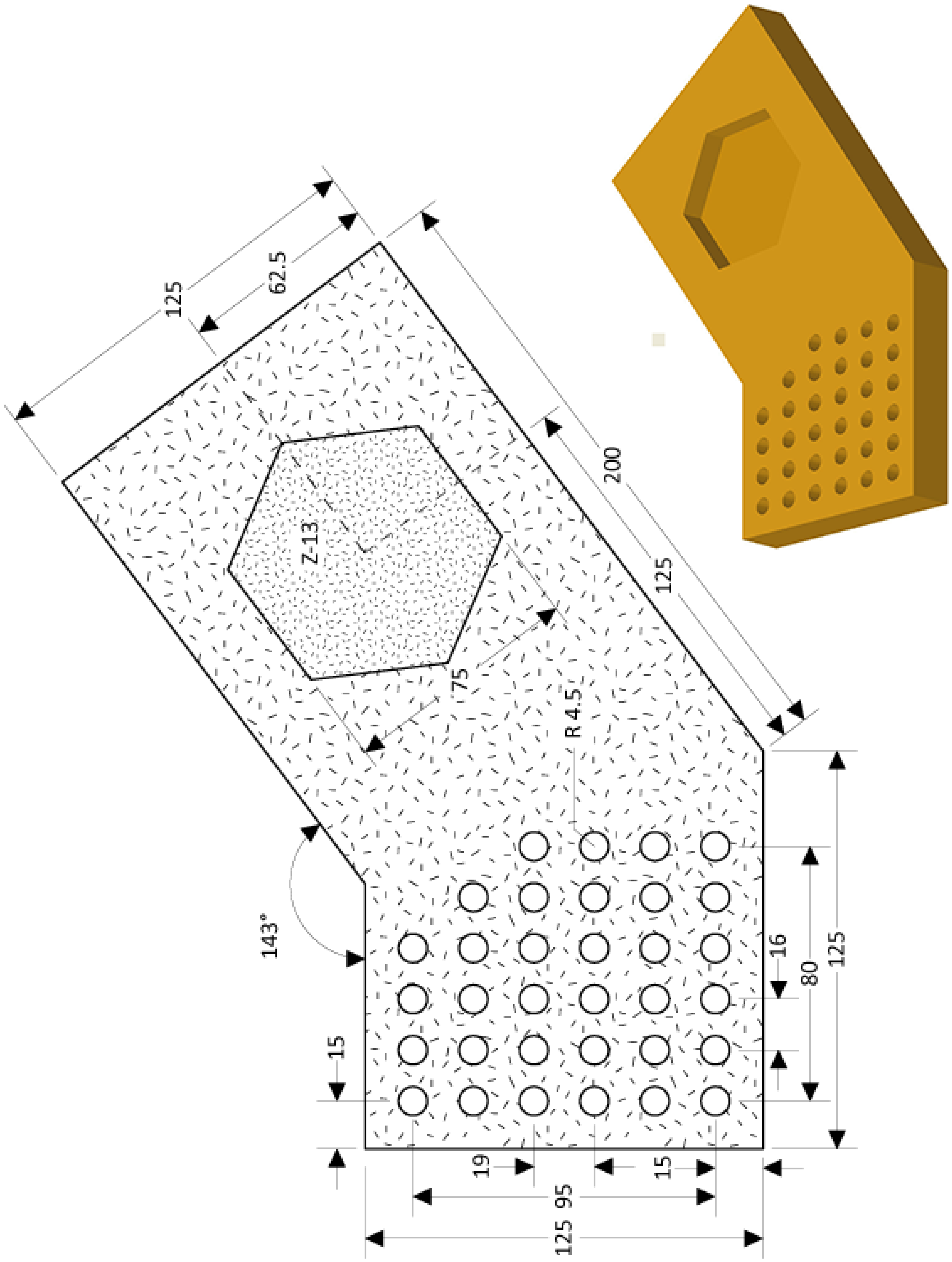


Top and Bottom

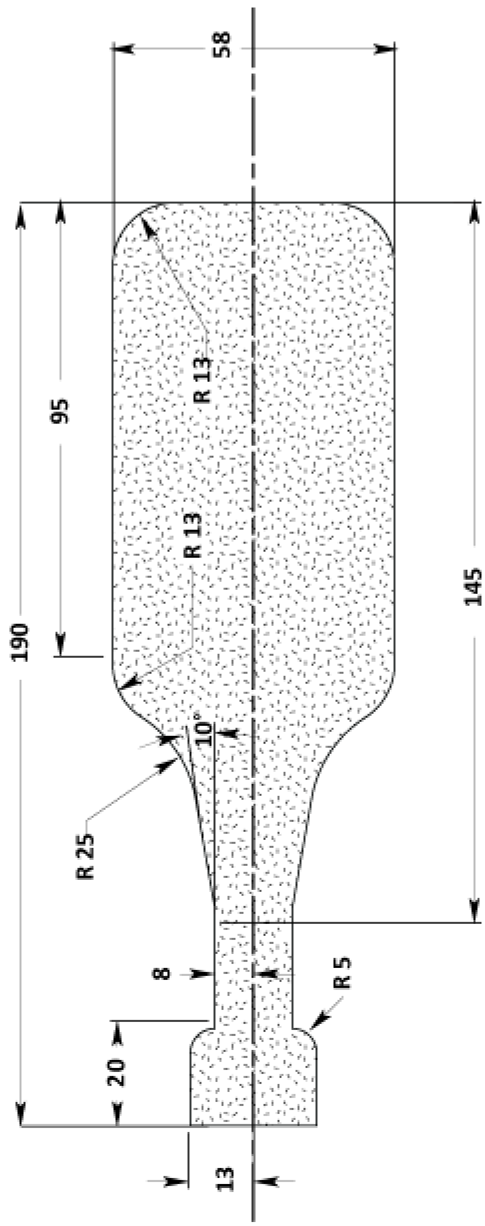




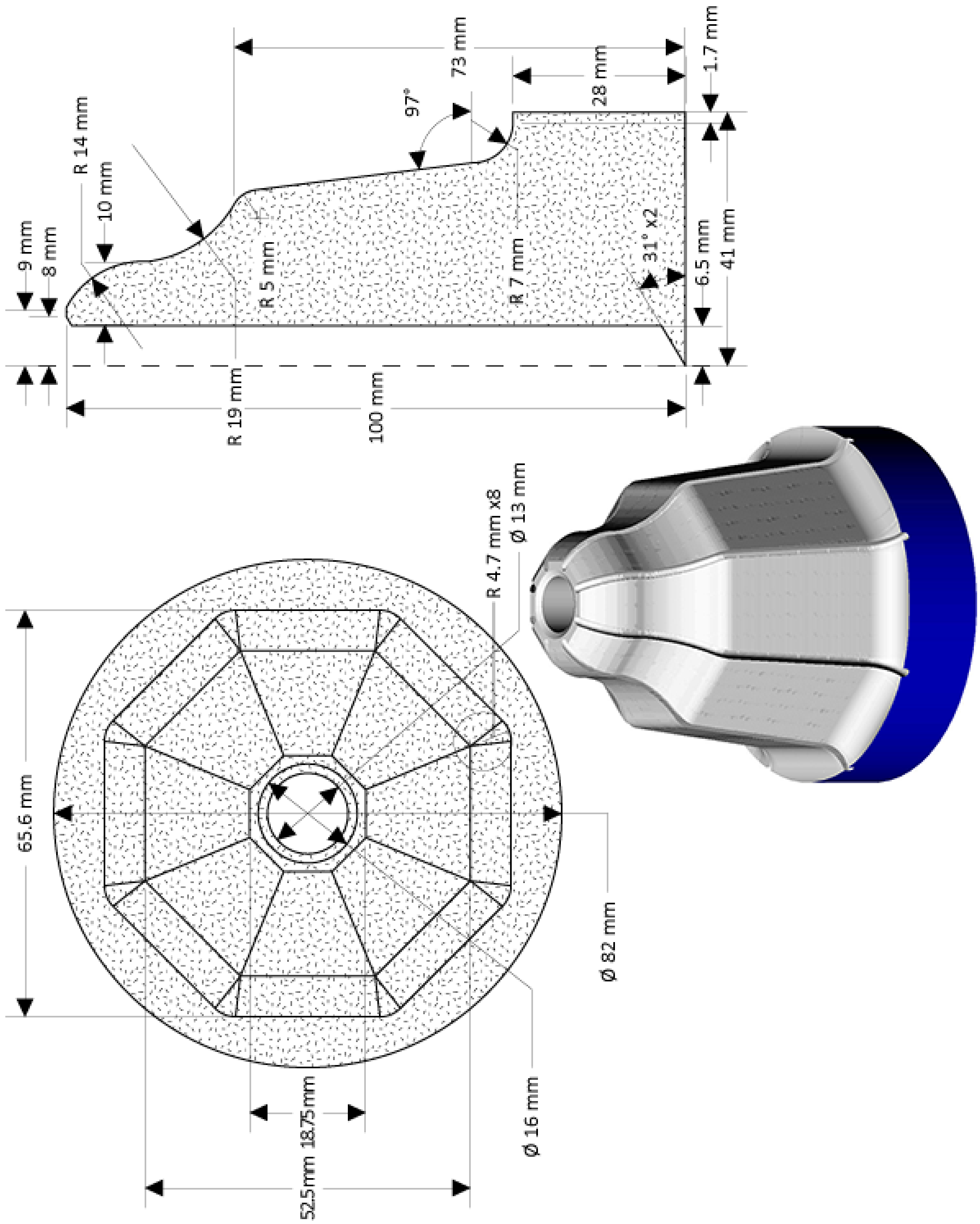
Multiple Origins



Bottle Mold



Z Machining


73 mm
28 mm
1.7 mm
97°
R 14 mm
10 mm
9 mm
8 mm
R 5 mm
R 7 mm
31° x2
6.5 mm
41 mm
R 19 mm
100 mm
R 4.7 mm x8
Ø 13 mm
65.6 mm
Ø 82 mm
Ø 16 mm
18.75 mm
52.5 mm