



Verisurf Cad Design Tutorial REV A3

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Exercise 1 -The Verisurf Interface

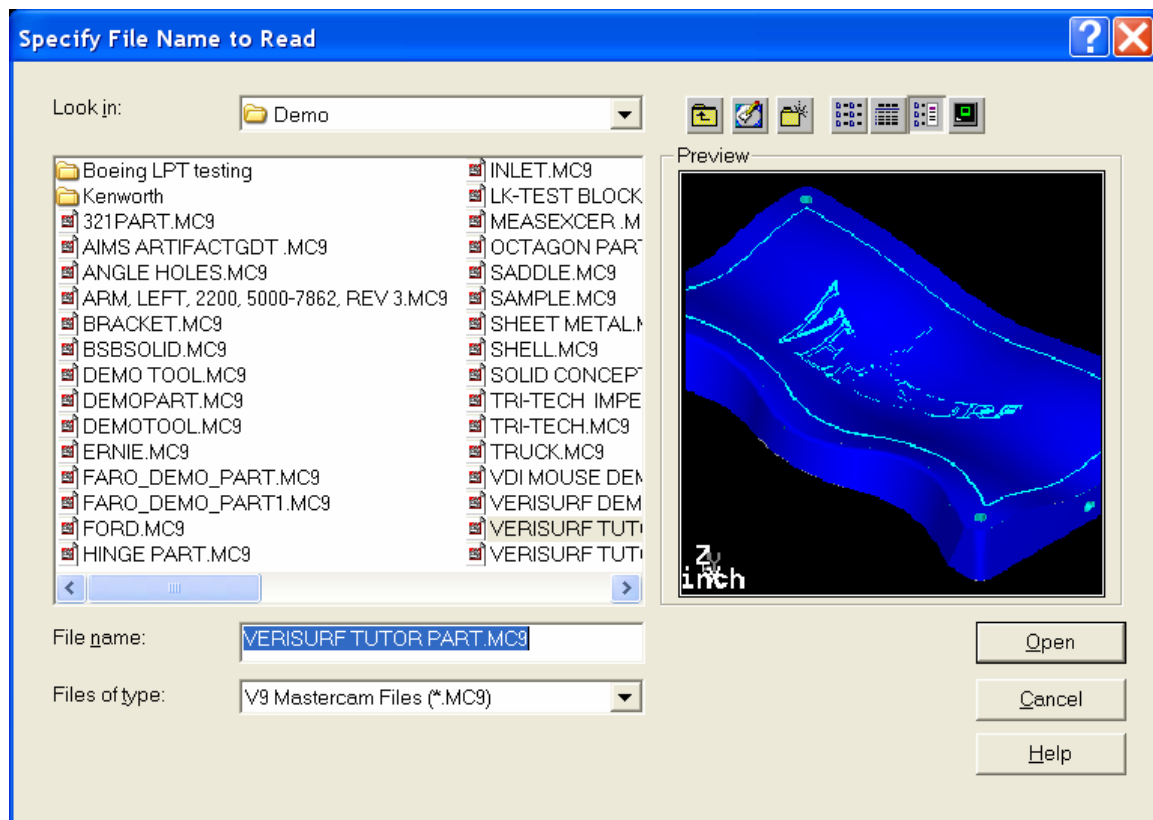
This exercise will utilize the items discussed in the CAD system basics, Learning the Verisurf Interface. You will learn the following skills that were discussed:

- Opening up a part file
- Gview, Zoom, Fit, Pan
- Color
- Level
- Z level
- Attributes

Before beginning, you should create a working folder to store all your parts as you work on them. C:\Mcam9\Design\MC9\Temp would be a suggested name.

Opening up a file

1. Choose the commands **File > Get**.
2. Go to the Demo folder and choose the part file named Verisurf tutor part.MC9.

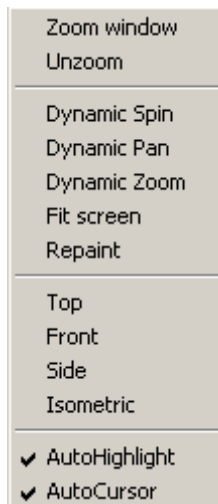


Gview, Zoom, Fit, Pan

This group of commands will show the student how to change the view, fit the part to screen and also move the part about the screen, or panning. These 4 commands should enable the new student to manipulate the part around the screen. Using all the commands shown below manipulate the part around the screen until comfortable.

- F1 – Zoom by window
- Alt-F1 – Fits to screen
- Arrow keys – Pan the part
- ALT + arrow keys – Rotates the part in 5 degree increments

In addition to those commands you can also right-click the mouse to access an array of movement commands.



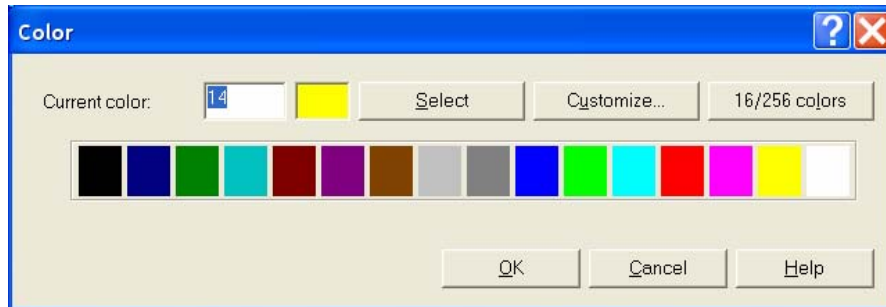
There are also some toolbar icons. The 10 shown are Zoom, Unzoom, Unzoom by 0.8 magnitude, Fit, Repaint, Dynamic rotation, ISO, Top, Front, Side



At this time go through the variety of choices and practice with all the types. Your instructor can answer any questions you may have on these commands. When complete , place the part in ISO view.

Changing the active or current color

1. Choose the **color** button **Color: 12** in the secondary menu. It shows that the current color is Red. Note that Red also has the number 12 associated to it. Every one of the 256 colors has a number associated to it.
2. Change the current color to Blue by clicking on the blue button. Note that the current color changes to number 14 and choose **OK**.



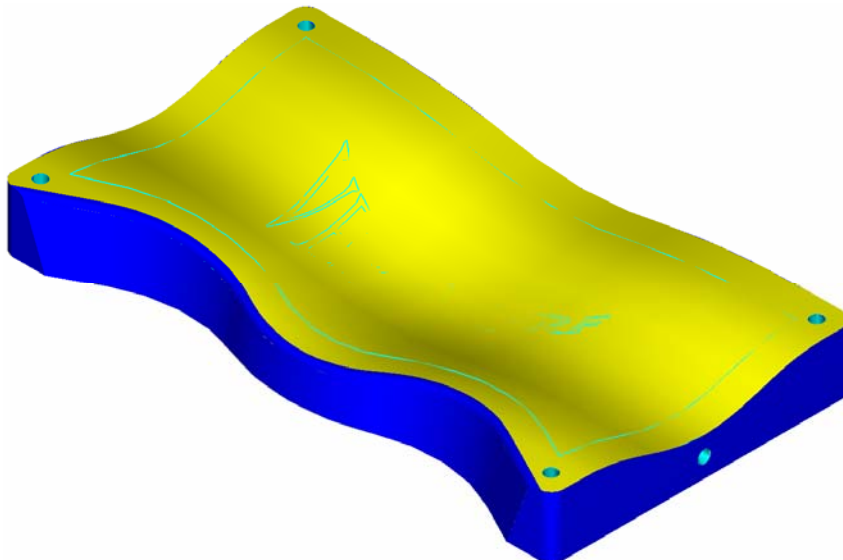
The color button changes to show the new color. From now on everything you create or edit will be in the new color.

Changing the color of an entity

1. From the main menu choose **Screen > Chg Colors**.
2. Move the cursor into the graphics area and choose the face. If you choose the wrong entity you can use the Undo button at the top in the toolbar.

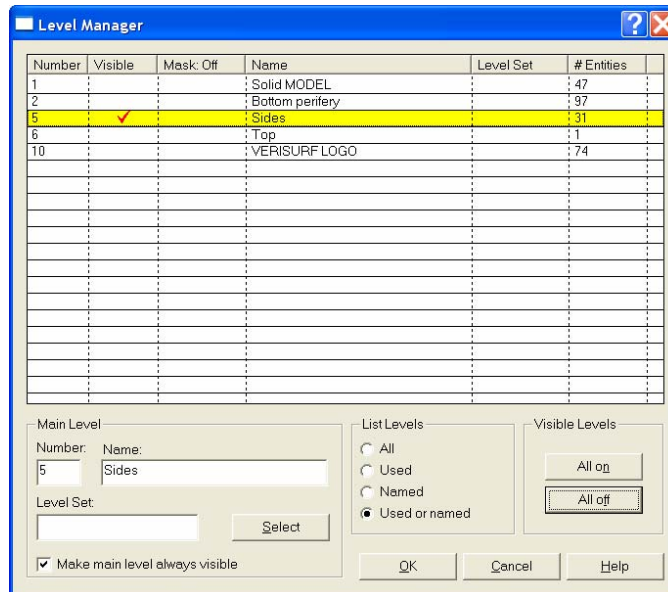


3. You should now use the cursor to get back to the main menu. Your part should look like this.

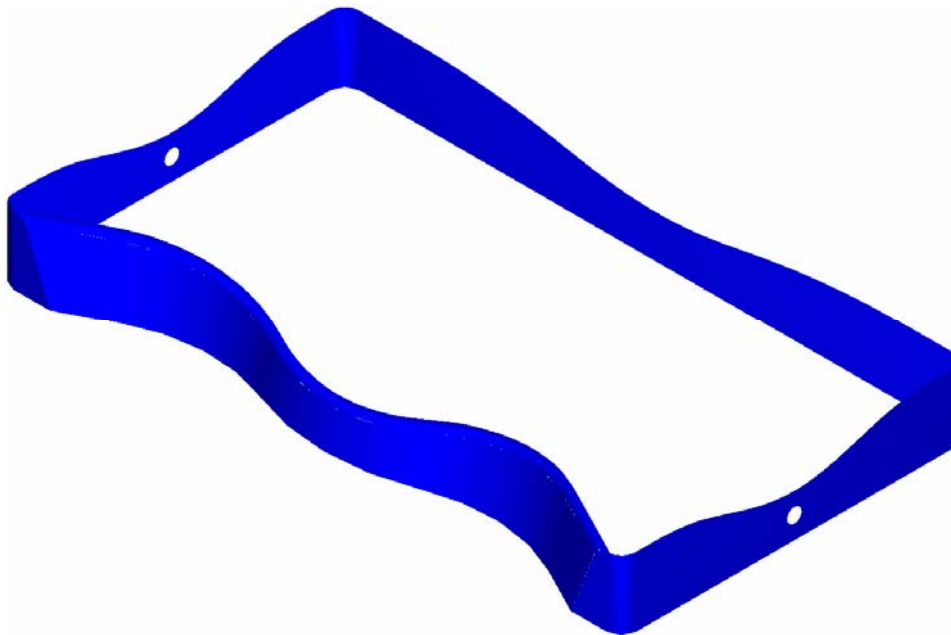


Levels and visibility of entities

1. Choose the level **Level: 2** button in the secondary menu. It will bring up the level manager.
2. Change the checkmark selections to resemble the ones in the illustration. You may have change the List Levels toggle to Used or named. This selection will leave only Level 5 visible.



Its 31 entities are the trimmed sides and the blanked ‘parent’ surfaces.




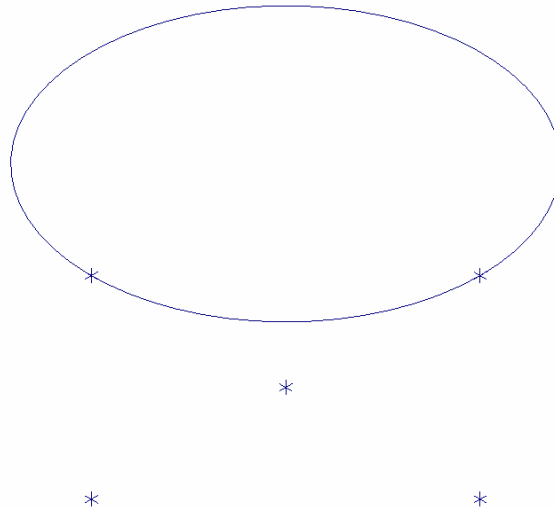
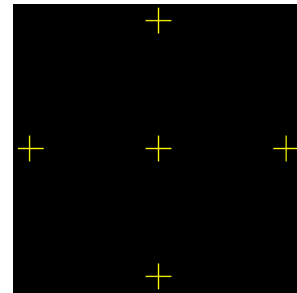
Changing color by level

1. Change active color to 10, Green.
2. Choose Chg Color > All > Level.
3. Choose level 5 (sides) > OK > Done.
4. All entities on level 5 should now be green.
5. Turn on all the levels again.

Z-level

To begin the tutorial on this topic we will need to learn some basic geometry building commands.

1. Go to the main level and choose **File > New**.
2. Change the Gview to Top. Color is 14, yellow.
3. Verify that Z is 0.0000 and Level is 1.
4. Create a point. From Main menu choose **Create > Point > Position**.
5. Type in 0,0 
6. A point is created at the origin.
7. Make 4 more points by continuing to type in the following X,Y positions: 1,0 and 0,1 and -1,0 and 0,-1
8. You should have 5 points as shown to right.
9. Now switch to an ISO view and change Z to 1.000.
10. This time we will make a circle.
11. At Main menu choose **Create > Arc > Circ pt+dia**.
12. Enter 2 for the diameter and then position the cursor on the point at 0,0 until the cursor locks on it. Hold the mouse there and notice that it looks like the surrounding 4 points will be on the circles circumference.
13. Click mouse and notice the circle went up along the Z axis. This is due to the Z being at 1.000. The circle is positioned at 0,0 but it is also at Z0.

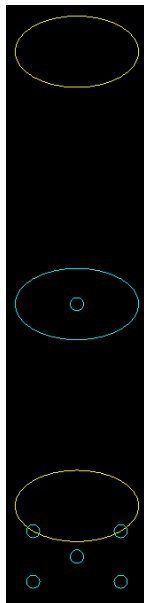
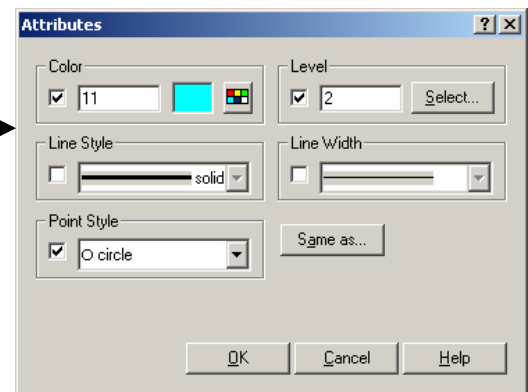
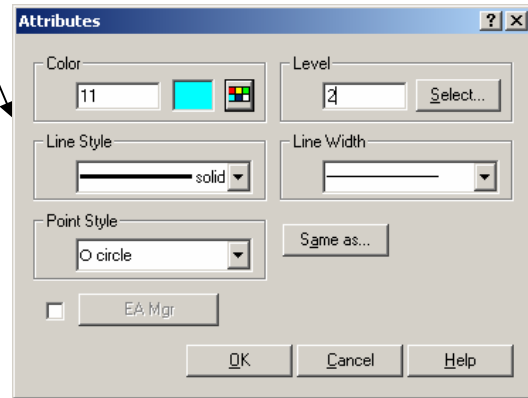


14. Make another Z change to 10.00 and create another circle using the 0,0 point for the centerline. You could also chose Origin from the main menu.
15. For an experiment go to main menu and choose Analyze. Go around and check all the entities for position. Now change Gview and look at the different views.
16. To see the Model Coordinate System press **F9**.

Attributes

Attributes is used to define what you entities will look like. In this exercise we will reset the attributes, create new entities and edit previously created entities.

1. Use the model from the previous exercise.
2. Choose the Attributes button in the secondary menu. **Attributes** .
3. Change the Attributes to match this.
 - Change the color to 11.
 - Change the level to 2.
 - Change the point style to **O circle**.
4. Choose **OK**.
5. Note that the secondary menu contains the changes.
6. Change Z level to 5.000
7. Create another 2.00 diameter circle at 0,0 or the origin.
8. Create a point at 0,0. Note the point is at Z level 5 and is a circle.
9. To edit the bottom 5 points to be on level 2, color 11 and point style circle. Go to Main menu and choose **Screen > Chg Attribs**. Checkmark the Attributes screen as shown.
10. Choose **Only > Points** from the entity selection menu.
11. Select the bottom 5 points.
12. Your ISO view should now look like this.



Alt-E Hide

The Alt-E or hide is a tool that you will find extremely useful when there are a large amount of elements in your model. To use it you key in Alt-E and then choose the elements you want to KEEP on the screen while the balance of elements are hidden.

1. Choose Alt-E.
2. In the selection menu choose All > Points and choose Done

Your model will look as shown to the right.

In conjunction with Alt-E you can add and subtract elements while the Hide is active.

3. Choose Alt- + (plus sign).
4. Select the circle at Z 1 inch and choose Done.

The circle is added to the elements shown on screen.

5. Choose Alt - - (minus sign).
6. Select the point at Z 5 inch and choose Done.

The point is removed from the elements on screen.

When you are ready to go back the original choose Alt – E again and you will be back where you started.

7. Open up the surface part model in the demo folder again and practice using Alt-E.

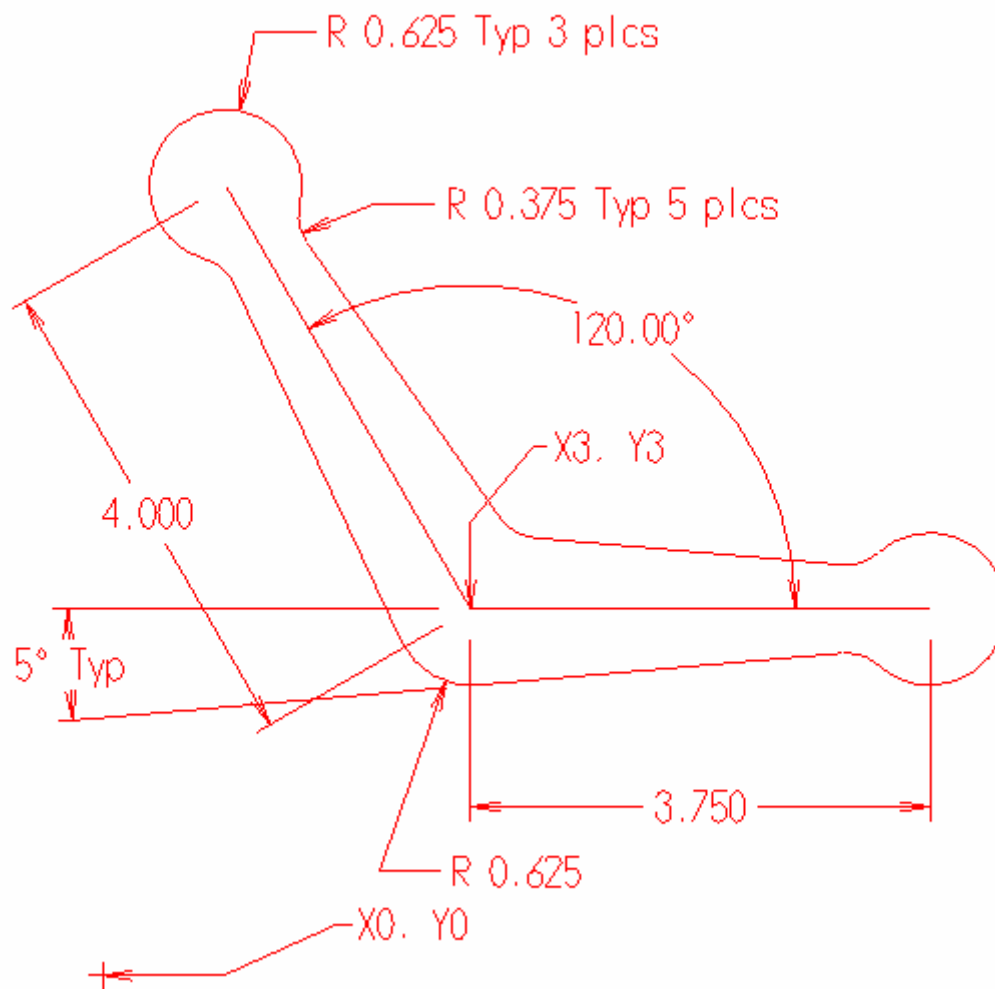


Exercise 2 – Basic drawing creation.

This exercise will introduce you to the major steps that typically go into making a part. This exercise is completed after reading the second section of the Cad reference manual. It will add a few new items too. This exercise will cover:

- Creating a new file
- Creating points, lines, arcs and fillets and splines.
- Mirroring and rotating lines
- Trimming lines and arcs.

The following drawing shows you what you will create.



Creating a new file

If necessary, follow these steps to create a new, blank Verisurf drawing.

1. Choose **Main Menu > File > New**
2. Choose **Yes** when prompted to create a new drawing.
3. If the current file has had any changes made to it since the last time it was saved, you will be asked whether or not you wish to save it. Choose **Yes** again if you wish to save it.

Creating construction guides

The first step is to create some construction guides to properly locate and orient the drawing.

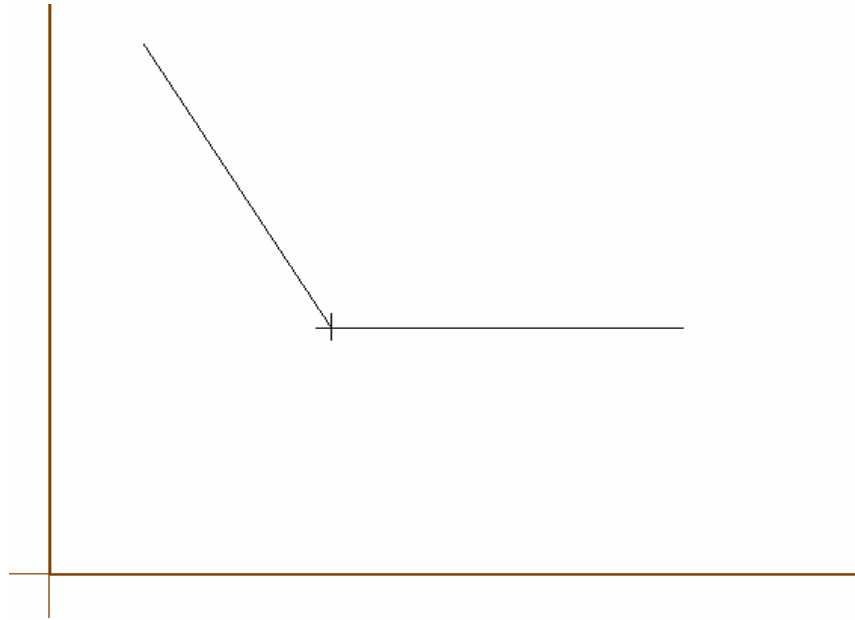
1. Create the center point of the elbow. Choose **Main Menu > Create > Point > Position.**
2. Enter the coordinates **3,3**. As soon as you start typing, the numbers will appear in the prompt area.

Tip: Press [Enter] after entering data in the prompt area.

3. Choose the **Screen-Fit button** on the toolbar or **Alt-F1** to center the point in the graphics window.
4. Next, draw the centerlines for the two arms. Choose **Main Menu > Create > Line > Polar.**
5. Click on the point to select it as an endpoint.

Tip: Pass the cursor over the point, when a square displays autocursor has locked onto the point. Click the mouse button to select.

6. In the prompt area, enter an angle of **0**.
7. Enter a line length of **3.75**. the guide for the horizontal arm appears.
8. Verisurf automatically prompts you to select an endpoint for another polar line. Click on the same point at 3,3.
9. Enter an angle of **120**.
10. Enter a line length of **4.00**.
11. Press **[F9]** to show the construction origin and X0Y0 axes. The part should look like the following picture.

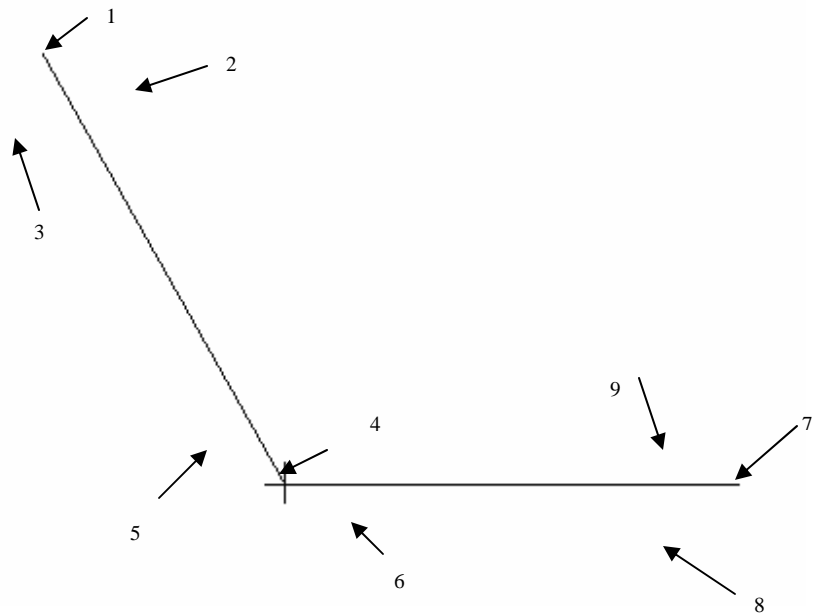


12. Press **[F9]** again to clear the axes from the screen.

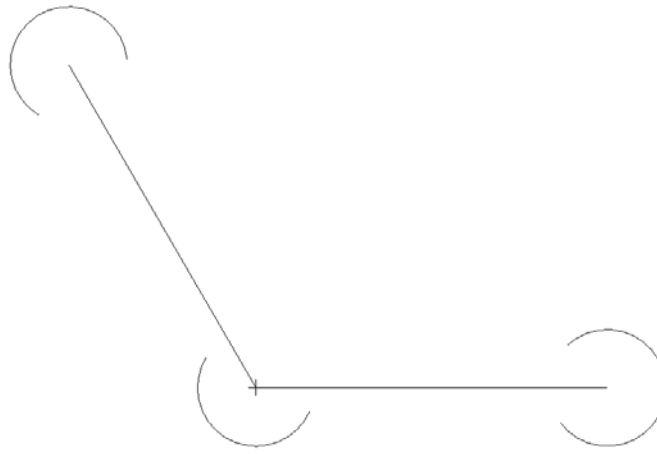
Drawing the arcs

Use arcs to create the rounded ends and curve at the elbows.

1. Choose **Main Menu > Create > Arc > Polar > Sketch**.
2. You are prompted to enter the center point for the first arc. Click on the line endpoint at position 1, as shown in the picture to the right.
3. Type the radius of the arc: **0.625**.
4. Specify the approximate ending position of the arc. Click at position 2 then at position 3.



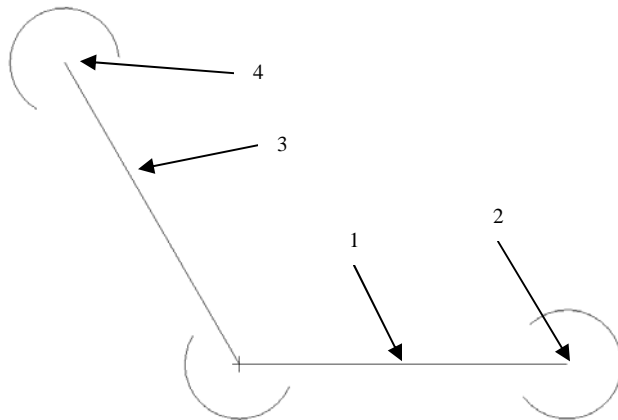
5. Repeat steps 2 through 4 to create the second and third arcs. Click on points 4, 5, 6 and 7, 8, 9 to create the other arcs.
6. If necessary do a **Fit Screen** again to fit the part completely to the screen. It should look like the following.



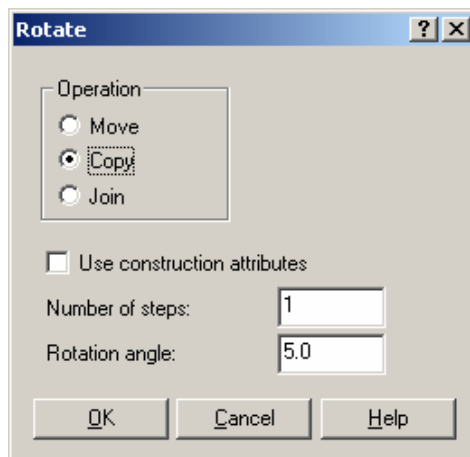
Rotating lines to create the arms

To create the outside edges of the arms, you will rotate the center line guides you created earlier.

1. Choose **Main Menu > Xform > Rotate.**
2. Select the line at position 1.
3. Choose **Done.**
4. Select the endpoint at position 2.



5. Enter the values shown in the following dialog box and choose **OK**. Use **5 degrees** for **Line 1** and **-5 degrees** for line 3.



6. Select the line at position 3

7. Choose **Done**.

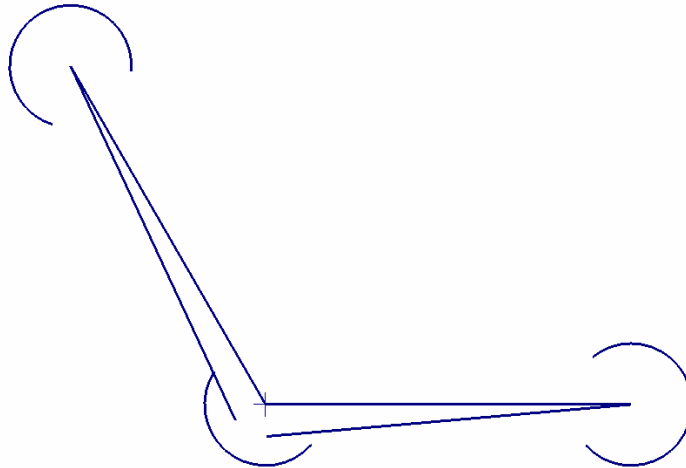
Tip: Instead of choosing menu items with the mouse, you can type the shortcut letter.

For each item on the menu, the shortcut letter is underlined.

8. Select the line endpoint at position 4.

9. Choose **OK**.

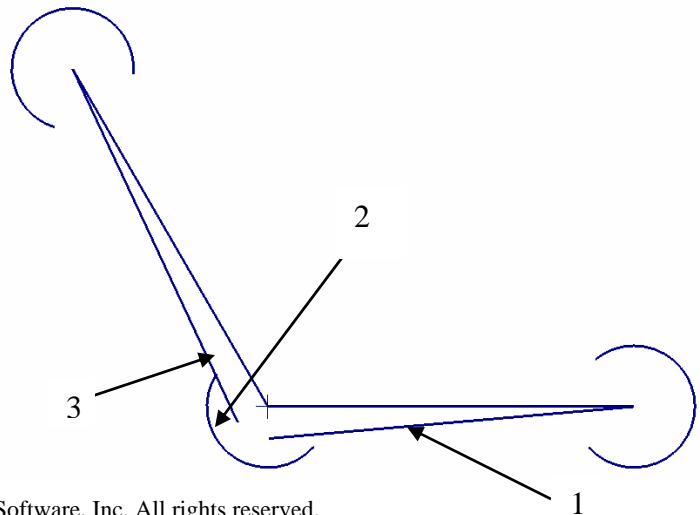
10. Whenever you do a Xform operation, Verisurf changes the colors of the original geometry and the new geometry so you can clearly see the results of the operation. Choose the **Screen-Clear colors** button on the toolbar or **Main Menu** > **S** > **L** shortcut to return the lines to their original color. The part should look like the following picture.



Moving the lines to the proper position

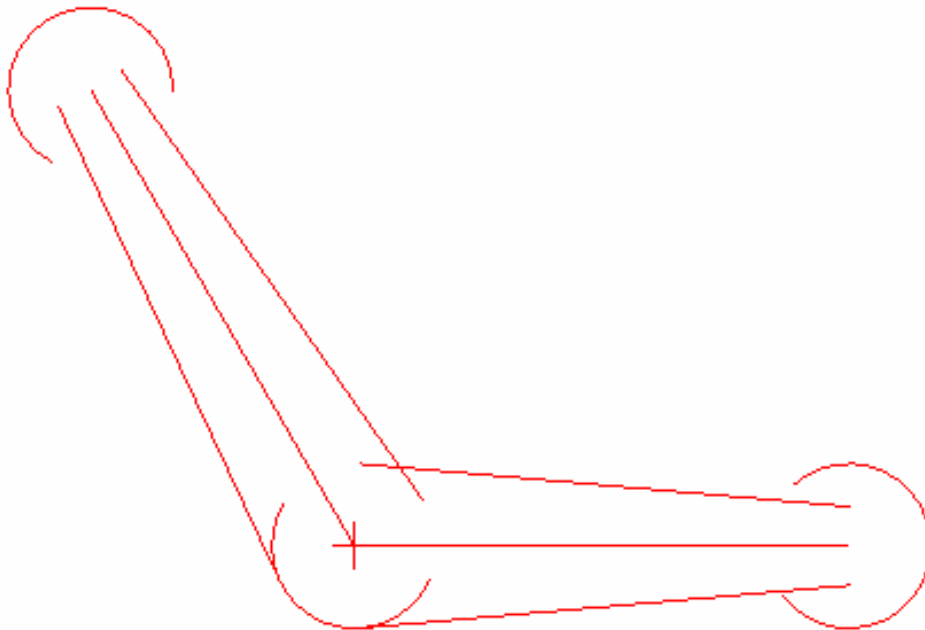
You have created the lines at the proper angle and orientation. Now, you need to move them to the proper position tangent to the arcs.

1. Choose **Main Menu** > **Create** > **Line** > **Parallel** > **Arc**.
2. Select the line at position 1.
3. Select the arc at position 2.
4. Verisurf shows you two possible lines. Click on the top line to keep it.
5. To create the second line, select the line at position 3.
6. Select the arc at position 2 again.
7. Click on the right line to keep it.





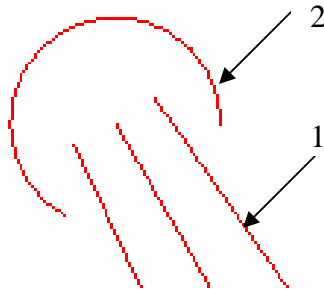
8. Choose the **Delete** button from the toolbar.
9. Click on lines 1 and 3 in the previous picture.
10. Finally, create the other side of the arms by mirroring the lines around the construction guides. Choose **Main Menu > Xform > Mirror**
11. Select one of the outer edges of an arm
12. Choose **Done**.
13. Select the corresponding construction guide.
14. Choose **Copy** and **OK** from the Mirror dialog box.
15. Repeat steps 11 through 14 for the other leg.
16. Clear the screen colors. Your part should look like the following picture.



Creating the fillets

Now we will create fillets to join the lines and arcs. You will also see how Verisurf can automatically trim lines to the base of the fillets. Complete the part by removing the remaining construction guides.

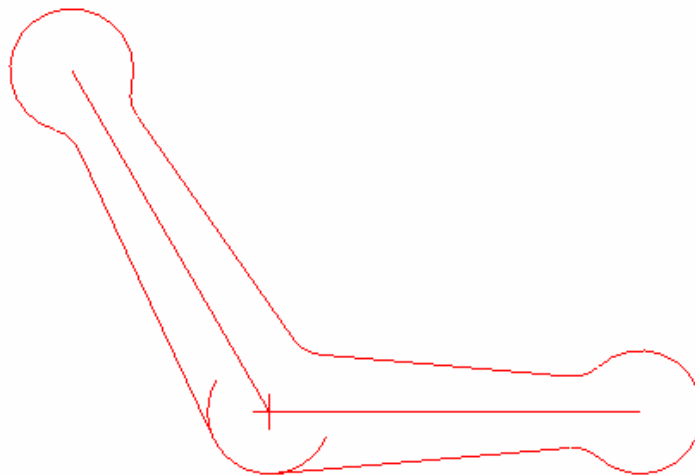
1. Choose **Main Menu > Create > Fillet > Radius**.
2. Enter the fillet radius: **0.375**.
3. Select the line at position 1 and the arc at position



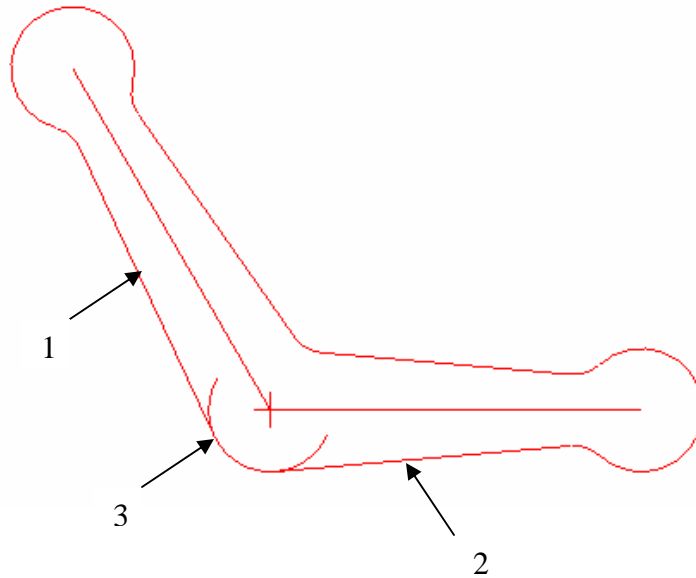
4. The fillet should look like the following picture.



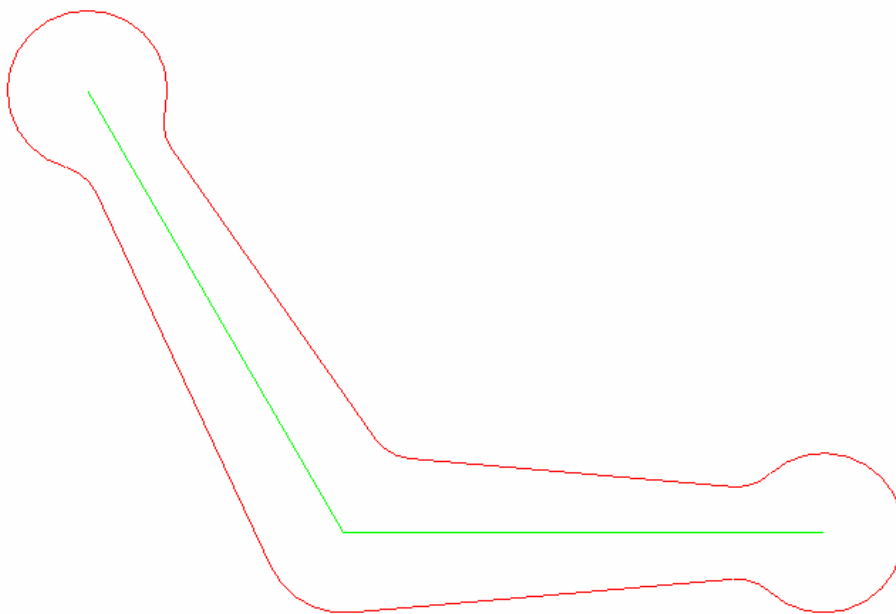
5. Select the lines and arcs for the remainder of the part to fillet and trim the balance. There are 4 fillets to yet create. Don't forget the one at the elbow. Your part should look like the following picture when done.



6. Trim the last arc to the adjoining lines. Choose **Main Menu > Modify > Trim > 3 entities.**
7. Select the lines at positions 1 and 2 (keeping these). Then select the arc at position 3 (trim this one).

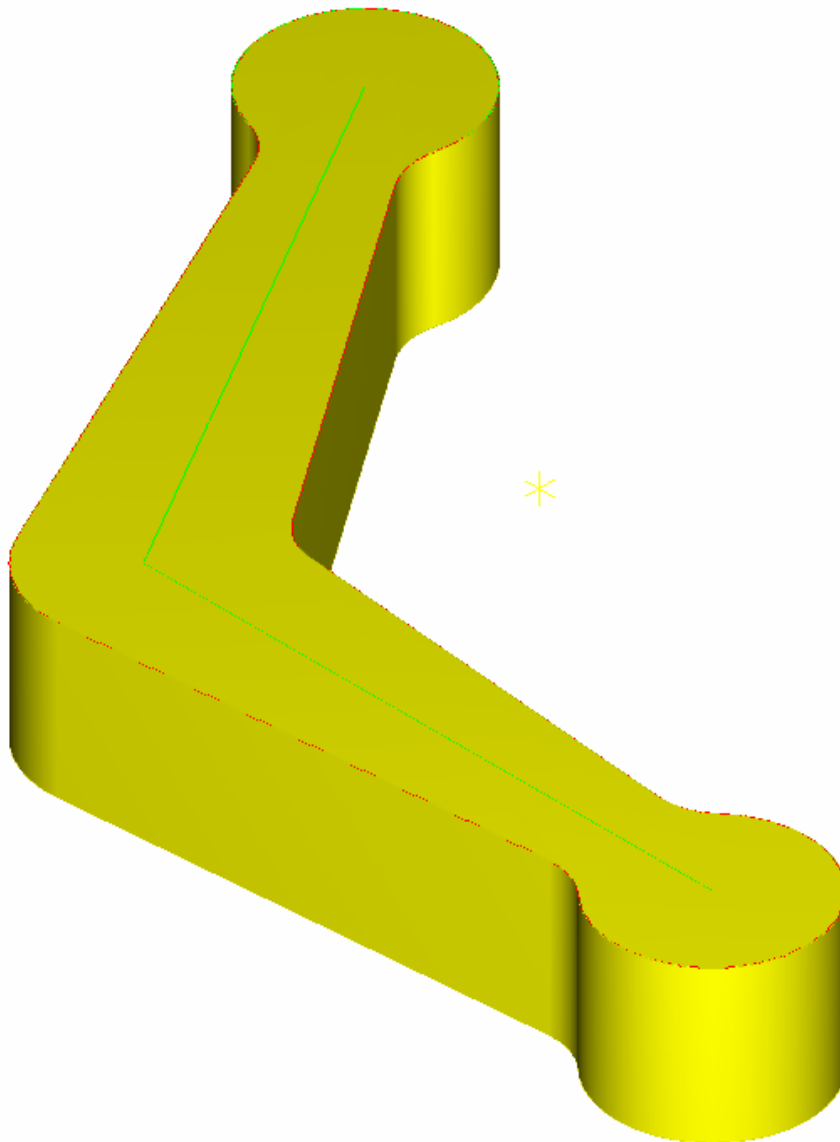


8. Your part should now look like this.



Extruding the model

1. Go to ISO view
2. Fit to screen
3. Choose **Main Menu > Create > Surface > Next Menu > Primitive > Extrusion.**
4. Select the elbow chain, it should light the whole chain of entities.
5. This will create surfaces around the chain.
6. Verify Extrusion Height is -1 in the lower left corner. If not change this by choosing **Height.**

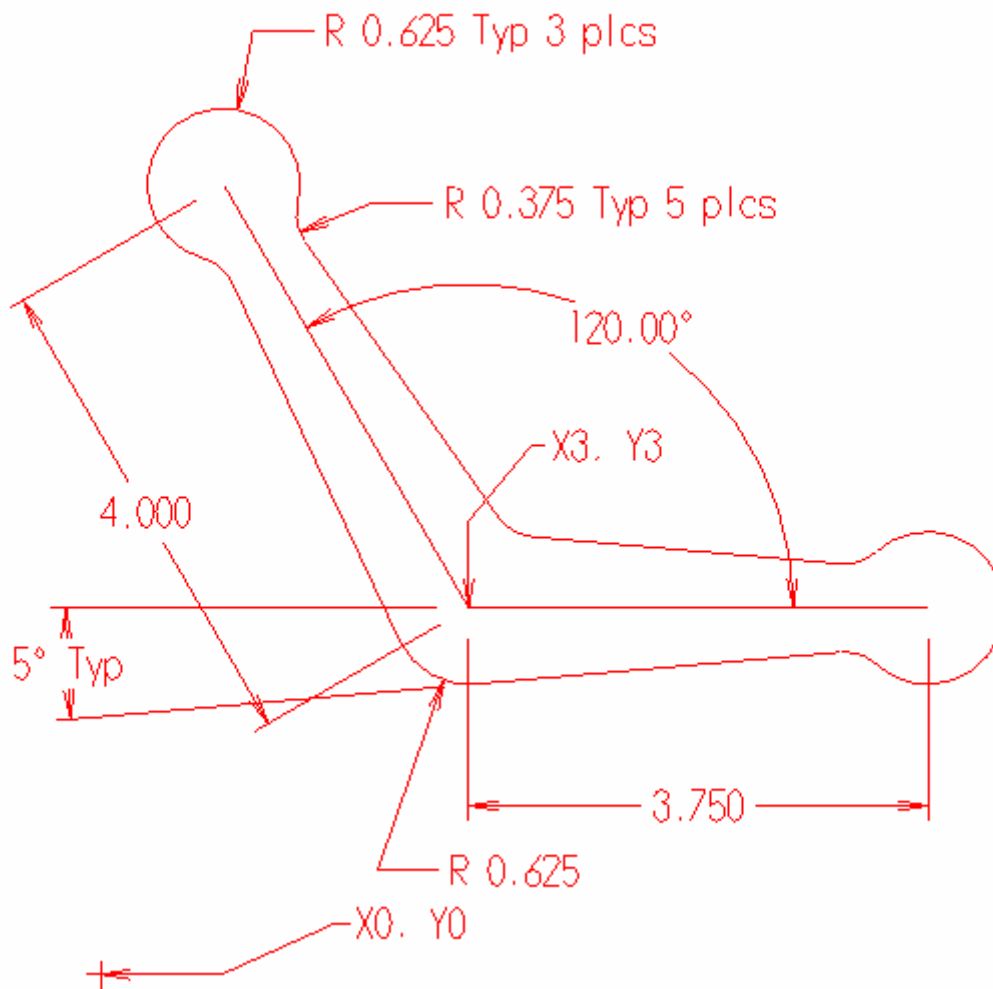


Adding drafting to a drawing

If you complete the part before others have you can add some of the drafting shown in the picture below.

1. Make a new Level 2 and name it Drafting.
2. Choose **Main Menu > Create > Drafting**.
3. Choose the 2 construction lines one at a time to see how easily it is to add the length drafting.
4. Choose the radii. One of each and place dimension.
5. Choose the 2 construction lines towards the ends to create the 120 degree callout.

If time allows the instructor will manually take you through the balance, however, drafting is only a small part of the training.




Exercise 3 – Manipulating views in Verisurf

To facilitate this exercise we will use the model from the prior exercise.

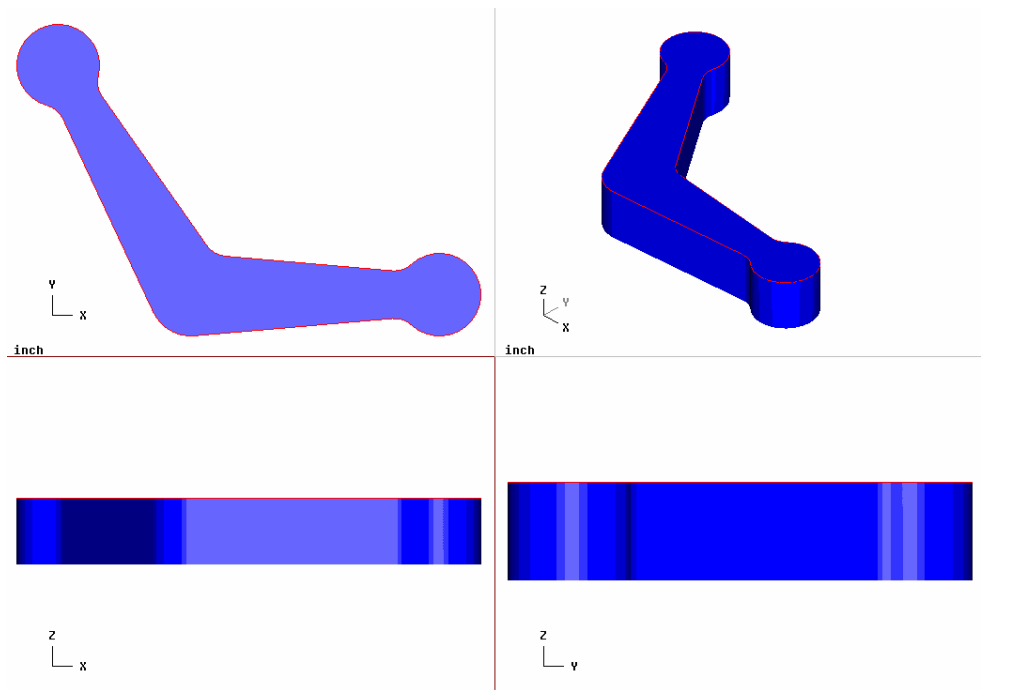
Using Gview


We will use both the Gview secondary menu button and the toolbar icons to change views.

1. From ISO view go to Top view using the Gview button .
2. Now change to Front and then Side.

Note that changing views also changes the Cplane (construction plane) settings but does not affect the Z-level. The Z-level is where any new geometry would be located parallel to the new Cplane.

Here are the 4 views; from top left going CW, Top – ISO – Side – Front.



3. Now use the toolbar buttons to see all the views available. .

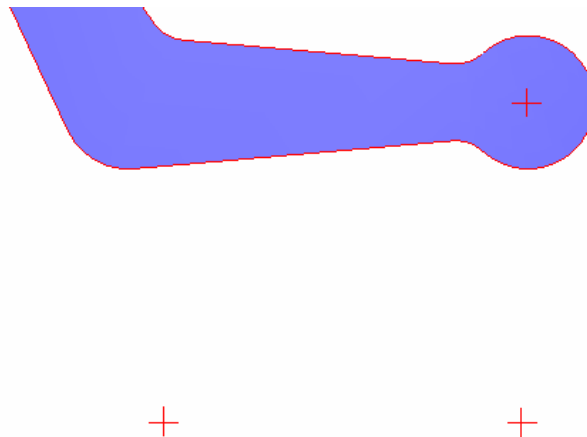
There are a few more buttons further to the right. Note that using these buttons implements a dynamic rotation to transition between the views.

We also see that the Z-level is still at zero. Again this indicates where any new geometry would be placed. To see the effect of the Z-level in conjunction with the Cplane we will make some points to better illustrate.

Creating entities in different Gviews

1. Change the **Color** so that the points will be a different color than the part.
2. Change **Attributes** point style to 3D star if needed.
3. Go to Top view. This should also set the Cplane to Top.
4. Create two points near the center of the bosses at the ends.
5. Manually rotate the part using the **Alt-arrow** keys to see that the points are physically on the top face of the part.
6. Turn on **F9** to see the home axes.
7. Use the toolbar to change to **Front view**.
8. Place two more points on the part anywhere.
9. Return to **Top** view and turn off the axes **F9**.

As you can see the points are on the Y zero. Test this by turning **F9** off and on.

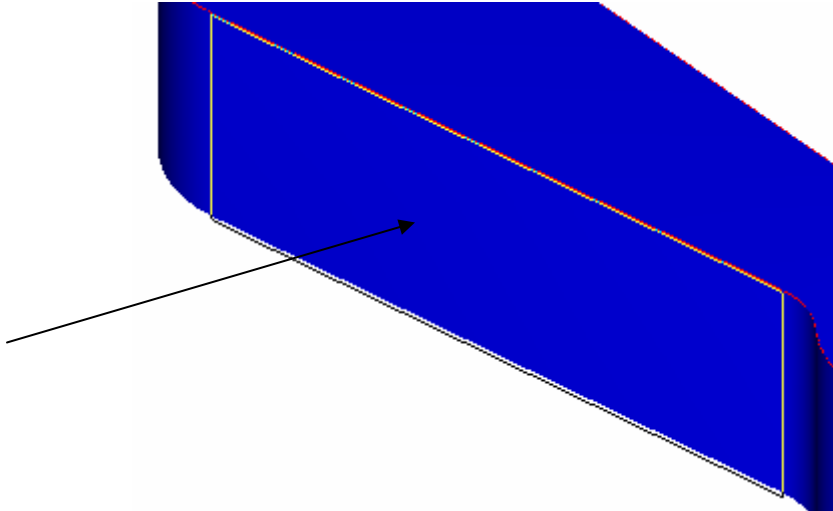


How then, if we are measuring parts, can we place the geometry in its proper space? The answers are in being able to move the Cplane to any angle in space and the Z-level to any position.

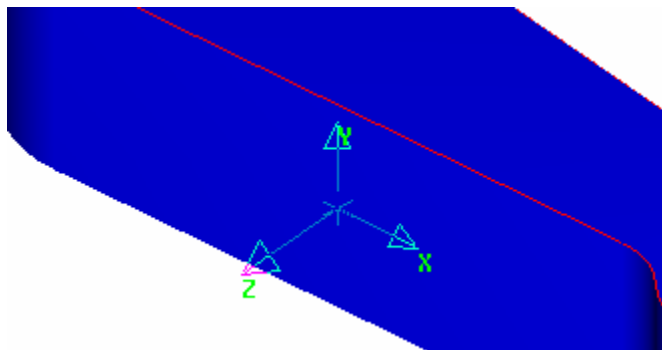
Creating entities in different Cplanes

In this phase of the exercise we will set the Cplane to be parallel to the faces of the sides of the elbow.

1. Go to ISO view.
2. Choose Cplane > Entity > Solid Face. Pick the face shown.



3. After choosing the face a Cplane coordinate system be shown for you to verify.



4. Choose Save from the menu and the CS disappears and a number appears in the Cplane secondary menu. This number is saved in the View Manager for future reference. Note that the Z-level has also moved.
5. While still in ISO draw a couple of points on the part on the face used as a Cplane. These points are now on the part.
6. As an experiment create an arc 2 inches in diameter and place one on the part and one off the part.
7. Rotate the part and note that both arcs are on the Cplane.

Shortcut Key Reference

Alt + 0	Verisurf Build
Alt + 1	Verisurf Measure Point
Alt + 2	Verisurf Measure 2D Line
Alt + 3	Verisurf Measure 2D Circle
Alt + 4	Verisurf Measure Plane
Alt + 5	Verisurf Measure Sphere
Alt + 6	Verisurf Measure Cylinder
Alt + 7	Verisurf Measure Slot
Alt + 8	Verisurf Measure Cone
Alt + 9	Verisurf Measure 3D Circle
Alt + A	AutoSave
Alt + B	Show/hide Toolbar
Alt + C	Run C-Hooks / DLL's
Alt + E	Hide entities
Alt + G	Selection grid parameters
Alt + H	On-line help
Alt + L	Set entity attributes
Alt + N	View Manager
Alt + P	Saves point data from Windows clipboard
Alt + S	Shading on/off
Alt + T	In Toolpath menu, turn toolpath display on/off
Alt + U	Undo last action
Alt + V	Mastercam version and SIM serial number
Alt + X	Color Manager
Alt + Z	Level Manager
Alt + `	Create two-point circle
Alt + +	Hide more entities
Alt -	Unhide Some
Alt + F1	Fit geometry to screen
Alt + F2	Unzoom to 0.8
Alt + F3	Cursor tracking on/off
Alt + F4	Exit Verisurf
Alt + F5	Delete using window selection
Alt + F7	Blank entities
Alt + F8	System configuration
Alt + F9	Display all axes
Alt + F10	Maximize and minimize screen
Alt + F11	VDI Temp
Alt + F12	Auto Align
Alt + arrow keys	Rotate Gview
F1	Zoom with window selection
F2	Unzoom
F3	Repaint
F4	Analyze entities
F5	Show Delete menu
F6	Show File menu
F7	Show Modify menu
F8	Home Tracker or Zero Arm
F9	Show/hide part information and coordinate axes
F10	List all functions and execute selected
F11	VDI Probe Mgr
F12	Feature Align
Esc	System interrupt or menu backup
Page up	Zoom in by 5%
Page down	Zoom out by 5%
Arrow keys	Pan